PIPING AND EQUIPMENT ANALYSIS & SIZING SUITE

Piping Systems Research
& Engineering Company
(NTP Truboprovod)

Equipment Stress Analysis and Sizing

PASS EQUIPMENT

Version 3.06

User's Manual

2023

Introduction

PASS/EQUIP is strength analysis software for vessels and their components designed for estimating load-carrying capacity in operation, test and assembly states. PASS/EQUIP is the basic module, which analyzes strength and stability of horizontal and vertical vessels using national and foreign standards.

PASS/EQUIP-Columns module analyzes strength and stability of columns considering wind and seismic loads.

PASS/EQUIP-Heat Exchangers module analyzes tube and casing heat exchangers (HE), including analysis of tube plates, tubes, pass partitions, casing, expansion joints, expansion vessel, floating head, and air–cooled heat exchangers.

Calculation of vertical steel tanks, designed for oil and oil product storage, is performed via module **PASS/EQUIP-Tanks**. For tanks, it is possible to create models of frame roofs and export it with loading and fixing to the ANSYS program for further strength and buckling calculations.

PASS/EQUIP-Seismic module analyses strength and stability of horizontal and vertical vessels considering seismic loads.

The program automatically creates a high-detailed solid model of vessel, with possibility of export to ACIS, IGES, Parasolid, STEP, JT, VRML, STL files.

This document contains a program overview, limitations of use, analysis methods, description of the user interface, information on required input data and analysis results, and installation and registration details.

A user-friendly interface and an easy to understand system for creating and analysing vessels makes the software accessible to any user. A convenient 3D graphic display allows easy verification of the accuracy of dimensions of both individual components and the entire model.

There may be small differences between this Manual's content and the software installed, as the program is being constantly updated.

Contents

1. General

1.1. Program overview

PASS/EQUIP software allows static and low-cycle fatigue strength and stability analysis of pressure vessels and their components to evaluate load-carrying capacity in operation state (including those operating in corrosive hydrogen sulfide environments), as well as in test and assembly states.

PASS/EQUIP analysis methods are based on the national and foreign codes listed in references.

Analysis is carried out for each component individually and includes the following:

- cylindrical shells (smooth and stiffened by stiffening rings);
- conical transitions;
- welded and bolted heads: spherical, ellipsoidal, torispherical, conical and flat (including reinforcing ribs and with a central opening) heads, spherical heads without knuckle;
- flange joints;
- nozzles in shells and heads;
- saddle supports and cylindrical shells for horizontal vessels;
- cylindrical shells and heads in areas of intersection with supporting lugs and legs for vertical vessels;
- cylindrical conical shells and dished heads at attachment points of lifting lugs, trunnions, joining pads; branches;
- bends;
- convex bulks;
- ellipsiodal transitions;
- column components under wind and seismic loads, including those mounted on a support structure;
- support shells of columns ;
- tube plates, casing, tubes, expansion joint, expansion vessel, floating head of heat exchange vessels;
- air-cooled chambers of heat exchangers, nozzles into chambers;
- jacketed vessel components(cylindrical, U-shaped, partially jacketed, coiled and half-pipe, with longitudinal pipes);

- components of high-pressure vessels (shells, heads: flanges, heads, stub-ins);
- components of vertical tanks;
- viewind windows, bosses;
- noncircular cross section (rectangular, oval, stayed and reinforced);
- strength and stability analysis of horizontal and vertical vessels is carried out considering seismic and wind loads.

PASS/EQUIP is recommended for design and verification analysis in oil-refining, petrochemical, natural gas, petroleum and other industries.

1.2. Capabilities

PASS/EQUIP basic module:

- data input and analysis. An error message will be display if all required data are not entered or data are entered incorrectly;
- input of additional wind, seismic loads, weight loads, forces and moments;
- thickness calculation (including for external pressure) and calculation of allowable pressure, forces and moments;
- analysis of vessel flange joints under pressure, external forces, moments and temperature stresses;
- automatic calculation of values such as weight, length, stiffening ring properties (in both cylindrical shells and saddle supports), circumference chord length, etc. after input of component dimensions and material properties;
- calculation of fluid volume, fill height, filling ratio and hydrostatic pressure in each component of horizontal and vertical vessel;
- calculation of volume and weight of the product in each insulated cavity of the vessel;
- representation of model structure as a structure tree.
- 3D graphic display which allows the color of separate components and the entire model to be customized;
- "wire-frame" and "transparent" view which allows internal components to be seen;
- displaying of model filling by product
- switching on/off of insulation and lining displaying;

- estimation of materials using;
- when dimensions or load properties of one component are changed, an option to automatically adjust all adjacent components is given;
- automatic creation of precise solid model of vessel and its export to popular solid modelling systems, i.e. ACIS, IGES, Parasolid, STEP, JT, VRML, STL.
- customization of measurement units;
- selection of materials from the database (as per GOST, ASME etc.) with possibility of adding new materials. Allowable stress, elasticity modulusand other values are automatically changed when changing material, temperature or wall thickness;
- selection of components from GOST (ATK) database (shells, heads, flanges, gaskets, studs of flange joints, saddle supports, supporting legs, cylindrical and conical supports, nozzles, cross-sections of ribs, stiffening rings, beam elements of support structure);
- analysis of horizontal vessel shells with any number (more than 2) and position of saddle supports; output of diagrams for deformation, bending moments, transverse forces and strength and stability allowances;
- calculation of many components (shells, heads, transitions) is performed as per the Russian (GOST, RD) and foreign (EN, ASME) codes selected by user.
- strength calculation of junction point between nozzle and vessel against influence of pressure and external loads, as per as per the Russian GOST 34233.3-2017) and foreign (WRC 537(107)/297) codes.
- calculation of pressure, external forces, moments and temperature stresses for valves and vessel flange joints, as per the Russian codes (GOST, RD), as well as per ASME VIII div.1 (pressure), ASME VIII div.2 (pressure and external loads).
- analysis of bolted heads (with flange joints) as a combined analysis of flange and bottom;
- calculation of low-cycle fatigue of vessel components;
- strength analysis of shells and heads considering displacement of weld joint edges, angularity and out-of-roundness of the shells;
- output, preview and printout of full (with intermediate analysis results) or short reports of component analyses;
- output of information on components that do not meet use or strength requirements;

- calculation of required test pressure as per components;
- building of compound unit model including two or more vessels;
- calculation of weights and positions of gravity centers, with consideration for filling per components and for vessel as a whole, in operating, assembly and test conditions;
- selection of thermal insulation of vessel components with consideration for climatic factors and work process-related parameters.
- export and import of vessel models from and to XML files;
- export of nozzles to Nozzle FEM format files (*.nzl);
- import of vessel models from MechaniCS XML format files.

PASS/EQUIP-Columns module:

- determination of vibration frequency and modes for column type vessels with any number of components, including support structure;
- calculation of forces under wind loads (including resonance vortex excitation) and seismic loads for columns ;
- strength and stability analysis of column components;
- analysis of “cylinder + cone” support with the option of including a connecting shell;
- automatic determination of position and properties of most unsafe cross-section of supporting shell;
- calculation of loads on basement and support structure (if any) of columns.

PASS/EQUIP-Heat Exchangers module:

- input of heat-exchange component properties within a single multi-window interface;
- calculation of forces in tube plates, casing and tubes;
- analysis of tube plates, casing, tubes, expansion joint, expansion vessel, floating head.

PASS/EQUIP-Tanks module:

- tank parameters setting in a single multiwindow dialog;
- automatic weight measurement;
- strength and stability analysis of the wall, stationary self-supporting roof and tank head, including wind, snow and seismic loads;
- creation of a frame roof model with automatic weight calculation;

- export of the model with loads and constraints to the ANSYS program for further strength and buckling analysis;
- wall anchorage estimation;
- calculation of loads on basement;
- estimation of allowable stresses on the nozzles of cut-ins in the tank wall;

PASS/EQUIP-Seismic module:

- calculation of loads from seismic forces on horizontal and vertical vessels of seismic resistance categories Is, IIs, IIIs;
- analysis of vessel components considering seismic loads;
- consideration of vessel installation height when calculating seismic loads.

1.3. Limitations of use

This software assumes certain limitations in the design of vessel components, which are described in corresponding codes of standards listed in references [5].

If any of the conditions are not met for a given component, a warning message will be displayed and analysis for that component will not be performed. Analysis of other vessel components can be continued.

2. System Administrator Manual

2.1. System requirements

2.1.1. Minimum configuration

Pentium 4 processor

1 GB RAM

1 GB free space on hard drive.

Video adapter 1024x768 or higher, 16-bit (65535 colors) or higher.

Windows 8/10/11

Internet Explorer 5.0 or higher

Dongle drivers (come with distribution kit).

2.1.2. Recommended configuration

Intel Core i5 2 GHz or higher

4 GB RAM

Video adapter with OpenGL 2.0 hardware support, resolution 1280x1024x24

Windows 8/10/11

Internet Explorer 7.0 or higher

MS Word 2003 or higher

2.2. Distribution kit

1) Flash-disk contains the software installation package.
2) License Agreement.
3) Registration form (if purchased through dealer)..
4) Dongle(s) (one per purchased copy) providing protection against unauthorized access.
5) Software printed documentation.

2.3. Software installation

CAUTION: dongle must NOT be inserted in the USB drive during installation in order to avoid its corruption.

To install the software, follow these steps:

1) Log in as administrator.

2) Check if system clock is set correctly. Incorrect system time could make the dongle unusable.
3) Insert installation disk/flash.
4) Run ***SETUP.EXE.***
5) Follow the installation instructions.

During the installation process, you will be asked to define the location path of the software and enter the directory name for the Start Menu. In addition, dongle drivers are installed if necessary. Distribution kit also includes ***Acrobat Reader*** software (installed separately, by launching arce505enu.exe in acroread.505 folder).

6) Insert dongle in USB port. System reboot may be required.
7) Launch the software by clicking on the PASS/EQUIP shortcut or opening the PASSAT.EXE. file.

If PASS/EQUIP software was purchased through OOO "NTP Truboprovod" dealers, a registration form must be sent in to activate the dongle. After receiving a dongle update string, dongle update program must be run to activate the dongle (see 2.5).

2.4. Network key (dongle) installation

A dongle can be placed both on the server and on any network computer, including computer with installed PASS/EQUIP. If the dongle is supposed to be placed on computer with installed PASS/EQUIP, it shall be done as per i.2.3. This section describes installation of dongle on separate computer (server).

CAUTION! During program installation the dongle shall NOT be inserted in the port, to avoid its damage.

To install the network key (dongle):

1) Log in with rights of administrator.

2) Check system clock truth. Wrongly set system time may lead to impossibility of working with the dongle.

3) Install data storage device with distributive.
4) Run file Redistr\Sentinel\ SPNComboInst_7.6.9.exe.
5) Follow all the instructions of the installation program.

To complete installation of dongle driver, system reboot may be required.

6) Insert dongle in USB port

2.5. Software protection against unauthorized use

PASS/EQUIP software is protected from unauthorized use. While PASS/EQUIP is running, some modules check for presence of dongle and, if dongle is absent, the software is switched to demo mode.

There are two types of dongles: local and network (network dongle can be supplied when purchasing 2 or more licences).

Local dongle authorizes the software on the PC where the software is installed. It is possible to run several copies of the application on the same PC.

Fig. 2.1

A network dongle authorizes the software for use on any PC within the local network. Maximum number of copies that can be run at the same time is controlled by the dongle and is equal to the number of licences purchased. A network dongle can be inserted in any computer on the local network (for instance, the server). Dongle drivers must be installed on the same computer. Dongle software uses TCP/IP protocol for network access.

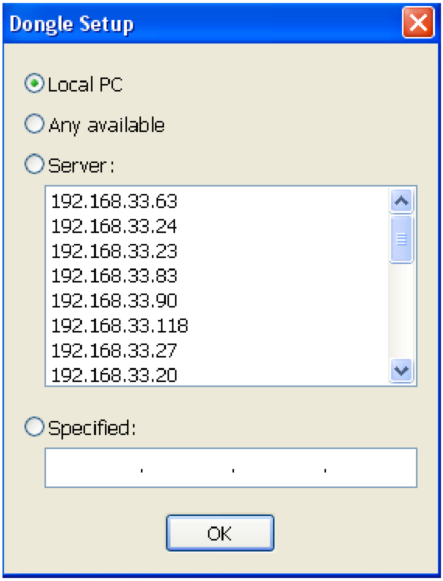


Fig. 2.2 Dongle setup

You can choose which dongle type the software will use:

- **local** – a local PC dongle will be used;
- **server** – a dongle on a specific PC (defined by IP-address) will be used;
- **Any available** – any available dongle will be used (local dongle will be chosen if present).

Dongle setup dialog box (Fig. 2.2) is opened by clicking "Setup" or selecting **Settings→Customize dongle access** from the menu. Local dongle is selected by default. If no dongle is available, software will run in demo mode. If a network dongle is placed on the PC where the sofware is installed, the PC's network address must be specified (“Local PC” setting cannot be used).

To activate dongle or change its settings in case of license renewal or a change in license details, use ***KeySt.exe***, which can be opened from **Start→Programs→Passat→Dongle Update.** A dialog box will open:

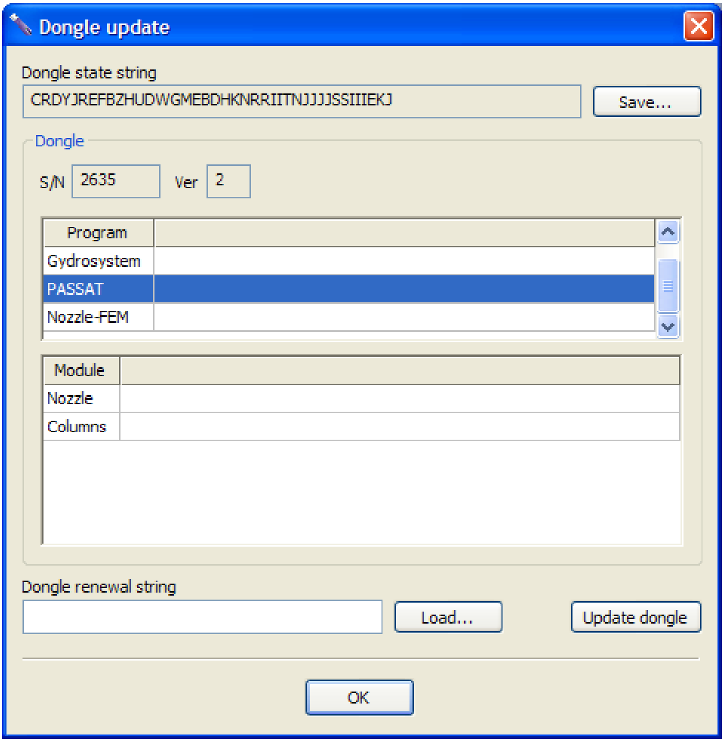


Fig. 2.3 Dongle update

To activate or update dongle, the code displayed in **Dongle state string** must be sent to PASS/EQUIP team. This code can be saved in a file by clicking **Save,** and then sent e-mail.

The code will be different every time the program is launched and can't be used repeatedly.

PASS/EQUIP team will provide a new code, which must be entered into the **Dongle renewal string**. The code can also be loaded from a file by clicking **Load**. When finished, click **Update dongle**.

You can check network key status, number of available licensed places and clients currently occupying a licensed place by running “Sentinel License Monitor”. “Sentinel License Monitor” can be run from a server computer by opening a web browser (e.g. Internet Explorer) and entering the url http://localhost:6002/ (default port is usually 6002). To run “Sentinel License Monitor” from another a client computer, “localhost” should be replaced with the name of the server computer. JAVA runtime environment installation may be required for the browser used in order to perform this operation (which should happen automatically if the computer is connected to the Internet).

2.6. Installation using Active Directory (AD) technology

Microsoft Windows Server 2003 and Microsoft Windows Server 2008 include integrated set of directory services Active Directory, constituent part of which Is Group Policy. Snap-in Software Installation, being part of Group Policy, provides remote installation of software on several workstations simultaneously.

Active Directory includes three (3) main installation scenarios:

- (Publish to User)
- (Assign to User)
- (Assign to Computer)

Attention!

- Software installation on the workstation will be finished only after reboot of the workstation.
- Software installation by scenarios Publish to User and Assign to User is not supported.

Software installation on the group of computers starts from creation of administrator setup. You can create this setup using ORCA MSI Editor. This program creates from file *.msi file setup.mst, which will save all changes introduced by administrator. Please find below the parameters, which are recommended to be adjusted before creating of mst-file:

Table	Parameter	Description
Directory	**INSTALLDIR**	Name of folder, where program files will be copied.
Directory	**SHELL_OBJECT_FOLDER**	Folder name in Start menu
Property	**Mode**	Dongle operating mode: 0 – local; 1 – any available; 2 – by specified address [**Server**].
Property	**Server**	Name of address of the server, which contains the dongle.

Property	**ProductLanguage**	Code of language, on which the program interface will displayed at the first start. By default 1049 (Russian)

Full program version is installed by default. Parameters Mode and Server inscribe dongle parameters into the branch of register HKEY_LOCAL_MACHINE\SOFTWARE\PSRE LTD\PassatXX\Settings.

After program installation, you should install dongle drivers for correct interaction with the dongle on local computers.

3. Working with PASS/EQUIP

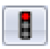
Software interface conforms to standards for Microsoft Windows applications and is based on standard dialog components of Microsoft Windows (menus, toolbars, dialog boxes, input fields, etc.); therefore, working with PASS/EQUIP should be intuitive for any Windows user.

3.1. Geometric kernel, solid model generation

Starting from version 2.08, the program uses geometric kernel C3D developed by C3D Labs Company. This kernel provides an automatic creation of high-detailed solid model of vessel (creation of reinforcing pads, fillets, cutting holes in the shells, etc.), and its export to popular solid modelling systems. The following formats are currently supported:

- ACIS
- IGES
- Parasolid
- STEP
- C3D
- JT
- VRML
- STL

However, creation of solid models also places additional demands on the system performance. With the lack of operating speed of the program, you can use ”Quick rebuilding of model”. Comparison of modes is indicated in the table below:

Icon	Mode	Function description
	Without rebuilding	Model rebuilding is not performed during changes. This mode is recommended for use with similar editing operations in the number of components.
	Accelerated model generation	In edit mode a solid model is built with some simplifications. Some components are displayed through OpenGL (bolts, heat exchanger tubes, trays, etc.). Reinforcement pads of nozzles are created by cylinder projecting, which at visualization may give significant distortions for tangential nozzles. For model rebuilding, an additional memory is required (500-1000 MB, depending on the model complexity). This mode is recommended for

		use for developing and editing of middle complexity models (50…100 components).
	Fine model generation	At editing, the solid model is built with maximum level of detail; reconfiguration may take a significant amount of time. All components are created as solid ones, with holes, fillets, etc. Reinforcement pads of nozzles are created by offset of the intersection line equidistant along the shell, which requires additional calculations. This mode may require additional memory (1-2 GB, depending on the model complexity). This mode is recommended for simple models editing.

Note: At calculation or export, the model will be automatically rebuilt in precise mode, if it hasn't been activated earlier.

3.2. Program model types

3.2.1. Horizontal vessels

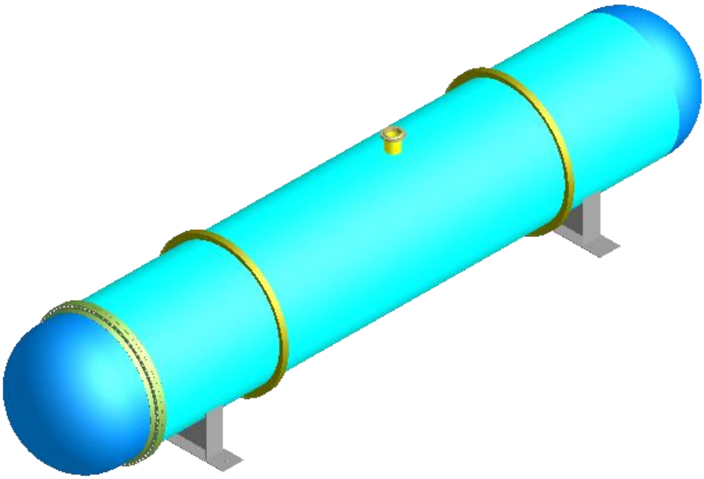


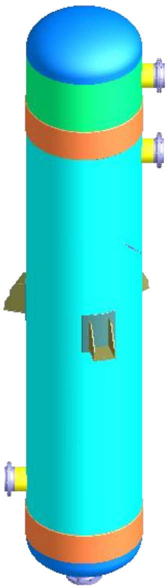
Fig. 3.1 Horizontal vessel model

Vessels of this type are usually installed on saddle supports. Model of horizontal vessel is formed from the components specified in i..3.9. Axis z is placed horizontally, along the vessel casing.

Loads calculation methods available for horizontal vessels:

Load		
Seismic	Wind	Snow
GOST 34283-2017	GOST 34283-2017	It is taken into account by the specific load on the service platforms (total pressure of snow, materials and other loads) or the load on the shell distributed along the length
GOST 55722-2013		
STO-SA-03-003-2009		
AzDTN 2.3-1 (AZE)		
IS 1893 (IND)	IS 875 (IND)	
EN 1998 (EUR)	EN 1991-1-4 (EUR)	
CFE 2015 (MEX)	CFE 2020 (MEX)	
ASCE 7-16 (USA)	ASCE 7-16 (USA)	
Inertial loads		

3.2.2. Vertical vessels



Vertical vessels are installed on the landing pads or leg supports of different types. Axis z is placed vertically, along the vessel casing.

Methods for calculating loads available for vertical vessels are similar to the section 3.2.1.

Fig. 3.2 Vertical vessel model

3.2.3. Column vessels

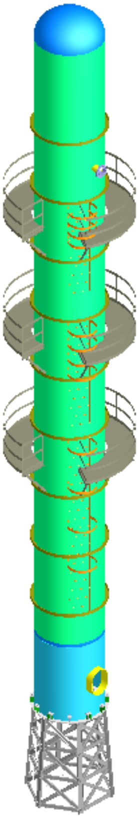


Fig. 3.3 Column vessel model

Vertical vessels, which are installed on the skirt support. For calculating of this vessels type, a license for “Passat-Columns” module is required.

Loads calculation methods available for column vessels:

Load		
Seismic	Wind	Snow
GOST 34283-2017	GOST 34283-2017	It is taken into account by

GOST 24756-81	GOST 24756-81	the specific load on the service platforms (total pressure of snow, materials and other loads) or the load on the shell distributed along the length
AzDTN 2.3-1 (AZE)		
IS 1893 (IND)	IS 875 (IND)	
EN 1998 (EUR)	EN 1991-1-4 (EUR)	
CFE 2015 (MEX)	CFE 2020 (MEX)	
ASCE 7-16 (USA)	ASCE 7-16 (USA)	

3.2.4. Vertical tanks

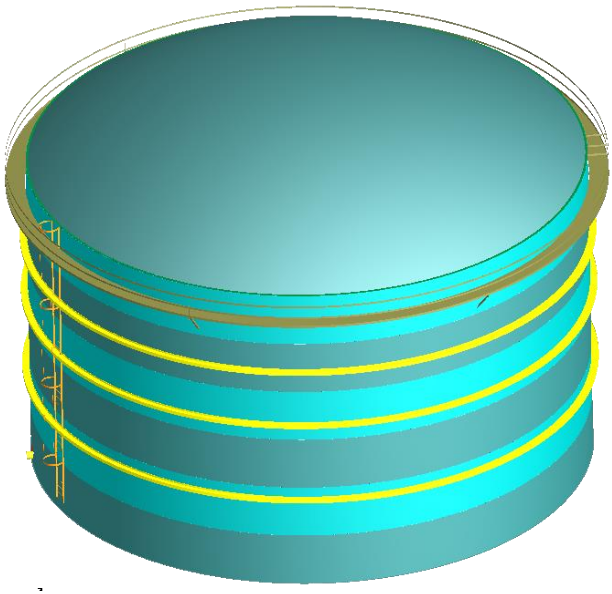


Fig. 3.4 Vertical tank model

Vertical filling tanks designed for storage of large volumes of product, with a flat head in the base.

For vertical cylindrical tanks a responsibility class shall be specified additionally, as well as snow area and turnaround of stored product with service life.

Loads calculation methods available for tanks:

Load		
Seismic	Wind	Snow
SP 14.13330.2014	SP 20.13330.2016	SP 20.13330.2016
AzDTN 2.3-1 (AZE)		
ASCE 7 (USA)	ASCE 7-16 (USA)	ASCE 7 (USA)
	API-650 (USA)	
CFE 2015 (MEX)	CFE 2020 (MEX)	
IS 1893 (IND)	IS 875 (IND)	
EN 1998 (EUR)	EN 1991-1-4 (EUR)	EN 1991-1-3 (EUR)

3.3. Creating, viewing and saving input and output data

Input data is in PASS/EQUIP format files with the following extension:

**.pst_horiz* – for horizontal vessels;

**.pst_vert* – for vertical vessels;

**.pst_col* – for columns.

**.pst_tank* – for vertical tanks.

The name of the active file is displayed in the title bar.

To create a new data file, use the **Create** command in the main menu or toolbar.

The new file will be create after it is saved for the first time. The **Save** command will work as **Save as** for a new file.

To save the active file, select **Save** in the main menu or toolbar.

To save the file with a new name, select **Save as** in the main menu. If required, an appropriate file will be created, opened and will become a current data file for the program.

When saving, a file type can be changed now, i.e. vertical model or column can be saved as horizontal model for calculations of tests in the horizontal position on the saddle supports. Not all of components can be saved in the new type of model, and appropriate notification will be displayed. If the mode type is changed, the saved file wouldn't become a current file.

To open an existing file, select **Open** in the main menu or toolbar.

To properly view images and mathematical equations, Internet Explorer must be set to display pictures(**Service→ Internet options→ Advanced→ Multimedia→ Display Pictures**).

3.4. Software window

The following window will appear when PASS/EQUIP is launched:

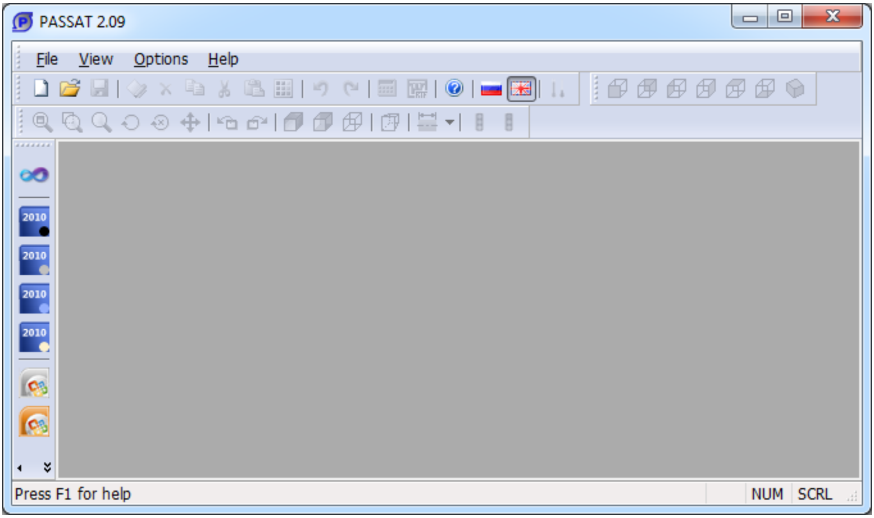


Fig. 3.5 Program window

To start, select **File** from the menu. Select **Create to open a new file or** Open **to open an existing one** (or use appropriate and icons).

A recently used file can also be opened from the **File** menu option. The number of available recent files is set in "**Document Options**" (see 3.11).

Vessel type must be selected before creating a new file (Fig. 3.6). Columns will be available only if **PASS/EQUIP-columns** module is licensed, or when working in demo mode.

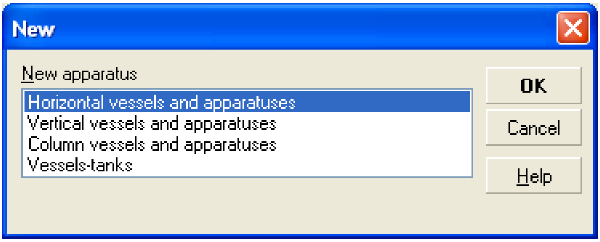


Fig. 3.6 Vessel types

When vessel type is selected or when an existing file is opened, a table containing general vessel data, data on the vessel's internal environment and analysis methods in test state will be displayed (Fig. 3.8).

After you press "OK", a work screen with a graphic view and toolbars containing all basic commands will be displayed (Fig. 3.7).

Graphic display of model is located in the middle of the screen. The symmetry axis of the model runs along the Z axis.

Icons in the right column of the screen are used for creating new model components.

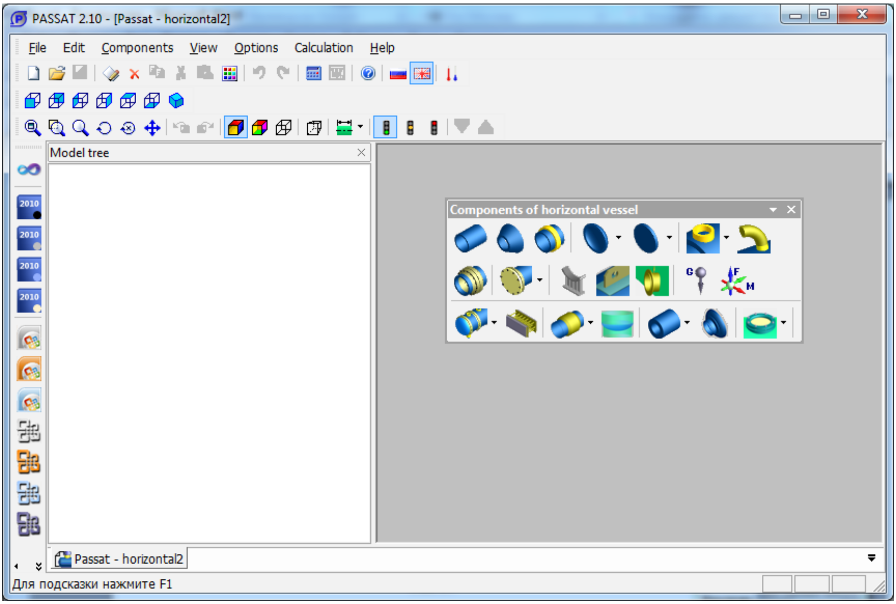


Fig. 3.7 Model window

When creating vertical cylindrical tank, a dialog for input data editing is opened automatically (i. 3.17.55).

3.5. General data

Dialog of general data includes main parameters of the vessel and its environment, information of its internal medium, types of calculation, etc.

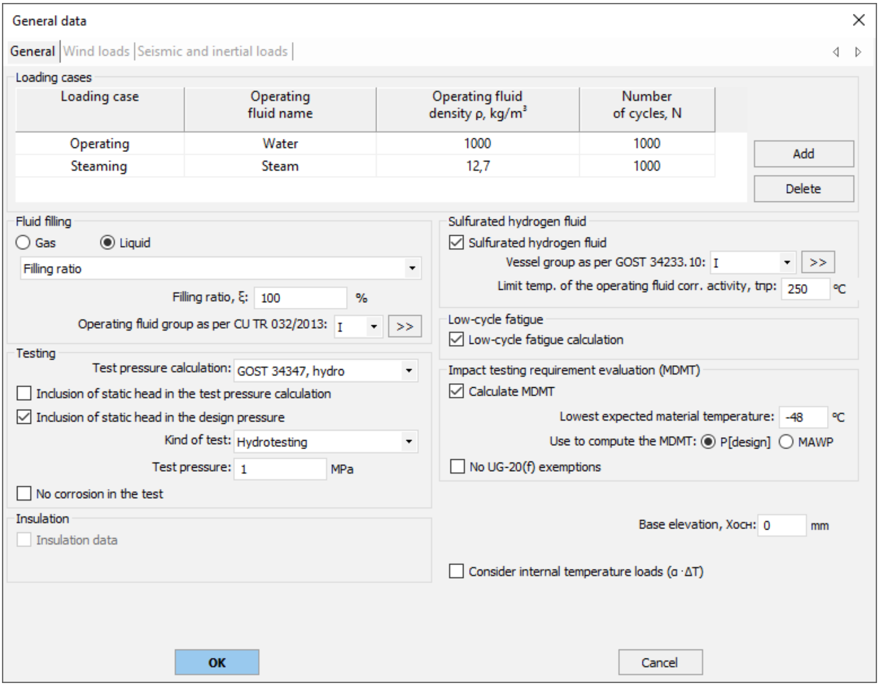


Fig. 3.8 General data

Parameter "Operating fluid group as per TR CU 032/2013” is designed to estimate the vessel category on the basis of this regulatory document.

If test state analysis is selected (in this instance "Hydrotesting"), all components will be analysed both for operation state and hydraulic test state with specified test pressure.

Liquid density and its fill factor (when using "Vessel carrying fluid") must be entered in order to calculate weight properties of vessel components in operation state. Vessel fill can also be set through fluid volume or fill height.

The “Load Cases” table allows to define several operating modes that will be simulated within one calculation. The loading case is characterized by the name, the name of the operating fluid and its density. The filling for all loading cases is considered uniform (to speed up the computing process), but the density of the operating fluid is assigned individually. This allows you to simulate various filling cases (working in steaming mode, etc.).

“Corrosion is not taken into account in the test calculation” enables to exclude a corrosion allowance (c_1) at calculations of all model components in test conditions, if they are performed for a new vessel.

"Calculation of test pressure" allows showing the code, under which a test pressure will be calculated.

“Inclusion of static head in the test pressure calculation” allows you to control subtraction of hydrostatic head pressure (pH), when evaluating a hydrotesting pressure. This item has appeared, because today there is no clear definition of the “test pressure” concept. In hydrotesting conditions, different components are subjected to different pressures (depending on the height of water). If, under test pressure, we assume pressure without hydrostatic one (“according to the upper gauge”), then in order to obtain pressure under hydrotesting conditions for a vessel component calculated in accordance with GOST, the test pressure shall be reduced by the hydrostatic pressure value. Otherwise, you can get an excessive pressure for the head.

“Inclusion of static head in the design pressure” allows you to control the influence of the static head on the design pressure (p) when evaluating the test pressure (ptest). This item has appeared due to discrepancies in the regulations regarding the determination of the design pressure (p).

“Hydrogen sulfide environment” must be selected when analyzing vessel components operating in corrosive hydrogen sulfide environments.

“Low-cycle fatigue analysis” must be selected for analyzing vessel components operating under cyclic loads where the number of cycles is between 10^3 and 10^6.

“Insulation calculation data” allows setting parameters, according to which calculation of thermal insulation for components will be performed (see. i.3.9).

“MDMT” (Minimum Design Metal Temperature) option allows you to assess the suitability of the material and the need for additional testing in accordance with the selected standard for each component. When this option is activated, a cell appears in which the user must enter the minimum temperature value at which the vessel can operate (based on the technological process or climatic data).

The option “Taking into account internal temperature loads” allows, when solving a beam model, to take into account loads due to thermal elongation of elements (with a rigidly clamped model or using non-standard fastenings).

3.6. Wind loads

The value of the natural period (T) is used in the calculation of wind and seismic loads for horizontal and vertical vessels. It can be calculated automatically or entered manually for each load case. The natural period of the vessel body is implied. Minor sections of the model (eg piping, internals) may have a lower oscillation period, but these values should be excluded from the analysis.

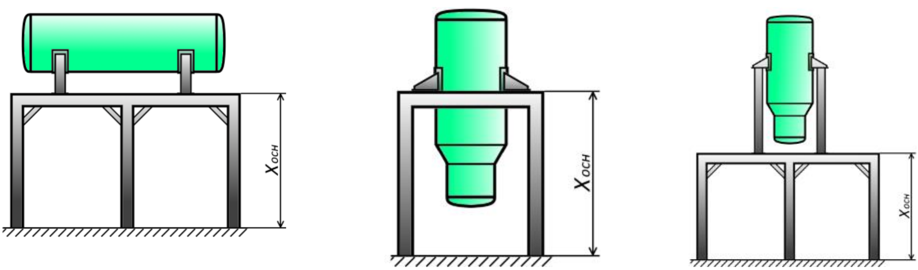
If "Calculation of vortex resonance" is selected, the chances of resonance and structure strength, if such resonance occurs, will be determined. This item is recommended for free-standing smooth and high structures, i.e. chimneys. In other cases, its activation may lead to excessively conservative assessment of strength.

3.7. Seismic and inertial loads

The "Allowance for seismic and inertial loads" option is required for calculating vessels taking into account loads due to seismic effects. This calculation is available for the PASSAT-Seismic, PASSAT-Columns, PASSAT-Tanks modules. It is necessary to select the standard according to which the loads will be calculated.

3.8. Elevation (height) of the vessel

This option provides consideration for the presence of any building structure under the vessel, which leads to an increase in wind and seismic loads.



Horizontal vessel on saddle supports | Supporting lugs | Supporting lugs welded poles

Fig. 3.9 Elevation of the vessel

3.9. Insulation calculation data

Specified properties are used for calculation of parameters of calculation of component thermal insulation (i.3.17.1.10)

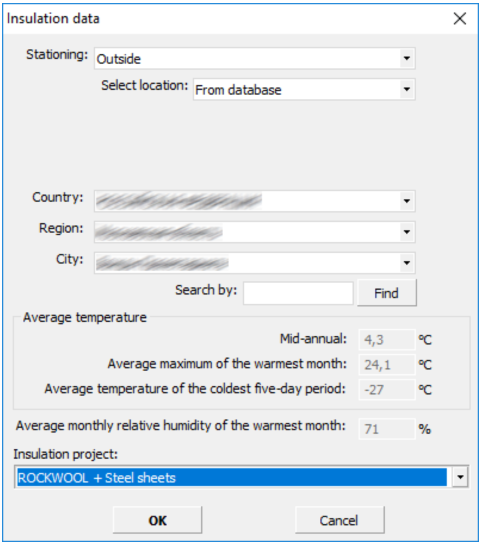


Fig. 3.10 Insulation calculation data

Parameter “Project of insulation” is the name of the package of rules, according to which insulation components are selected.

3.10. Main menu

The Table 3-1 briefly describes all available main menu items.

Table 3-1

Menu item name (icon)		Function
"File" submenu		
	New (Ctrl+N)	Create a new file
	Open (Ctrl+O)	Open an existing file (with *.pst extension)
	Close	Close active file
	Save (Ctrl+S)	Save active file
	Save as	Save active file with a new name
	Export to XML	Export model to XML file
	Export to Nozzle FEM	Exports nozzle data to a "Nozzle-FEM" file.
	Export to C3D, IGES, STEP, ACIS, ParaSolid, JT, VRML, STL	Saves solid model of vessel to one of formats
	Import from XML	Import model from XML file
	Import from XML MechaniCS	Import model from XML MechaniCS file
	Exit (Alt+F4)	Exit program
"Edit" submenu		
	Undo (Ctrl+U)	Cancel the last command
	Redo (Ctrl+R)	Redo the last cancelled command
	Edit (F4, double click)	Edit model components
	Delete (F8, Delete)	Delete model components
	Copy (Ctrl+C)	Copy selected component to clipboard. All component data except name will be copied as

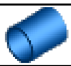
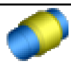
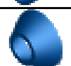
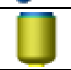
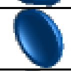
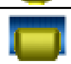
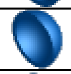
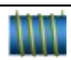
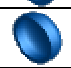
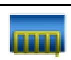
Menu item name (icon)		Function
		well. Sub-components will not be copied.
	Cut (Ctrl+X)	Similar to copy, but after being pasted, the original component will be deleted (after user's confirmation)
	Paste (Ctrl+V)	Paste copied component into the model. If an component is selected at this time, the new component will be adjoined to it. If no components are selected (or if selected component has several possible joint locations), you will be asked to specify the new component's joint location.
	Change color	Adjust display color of model components

Menu item name (icon)		Function

Menu item name (icon)		Function

Menu item name (icon)		Function
"View" submenu		
Toolbars		Turn on/off the following toolbars: View (3D model view options), Standard views (common vessel model views), Components (components which can be added to the model), Themes (change interface styles).
Settings…		Adjust interface and toolbars and set hotkeys.
Status bar		Turns status bar on/off.
"Options" submenu		
Units		Set measurement units used for dimensions, load properties and material properties.
General data		Display (and edit) general model data.
Customize dongle access		Displays network or local dongle settings

Menu item name (icon)		Function
Settings…		Displays software settings
	Language…	Change interface and output language (Russian or English).
	Components temperatures…	Provides setting of calculation temperatures simultaneously for several components of the model
	Components insulations…	Provides setting of thermal insulation parameters simultaneously for several components of the model
"Calculation" submenu		
	Vessel calculation (F3)	Run calculation and produce output file
	Converting to WORD (Ctrl+W)	Create output report in RTF format (MS Word).
"Help" submenu		
	Help	Open help file.
	Check for updates	Runs an integrated automatic update system.
	About Passat program	Display software version, support contact e-mail and copyright information.

Components library			
	Cylindrical shell		Cylindrical jacket
	Conical transition		U-shaped jacket
	Ellipsoidal head		Partially jacketed vessel
	Spherical head		Half-pipe coil jacket
	Torispherical head		Half-pipe battery jacket

	Flat conical head (α>70°)		Jacket with longitudinal channels
	Steep conical head (α≤70°)		Ellipsoidal bulk
	Spherical head without knuckle		Spherical bulk
	Flat head (cover)		Torispherical bulk
	Flat head with ribs		Virtual bulk
	Integral flat heads with opening		Ellipsoidal transition
	Oval head		Expansion bellows
	Nozzle		Heat Exchanger with stationary tubesheets
	Oval nozzle		Heat exchanger with U-shaped tubes
	Bend		Heat exchanger with floating head
	Saddle support [a)]		Air–cooled heat exchanger
	Bracket supports [a)]		Nozzle (tie-in) to air–cooled heat exchanger
	Stiffening ring		High pressure cylinder
	Stiffening rings group		Ellipsoidal high pressure head
	Flange joint		High pressure flat head
	Reversal flange		Spherical unbeaded high pressure head
	Bolted flat head		Bolted high pressure flat head
	Bolted ellipsoidal head		Bolted high pressure spherical head
	Bolted spherical head without knuckle		High pressure flange joint
	Bracket supports [b)]		High pressure bend

	Supporting legs [b)]		High pressure nozzle
	Supporting lugs [b)]		Packing [c)]
	Supporting legs on the shell [b)]		Service platform
	Supporting ring [b)]		Tray block [c)]
	Lifting lug		Skirt [c)]
	Joining pad		Viewing window in the boss
	Trunnion		Viewing window in the nozzle
	Lumped mass		Flanged boss
	External loads		Vessel assembly
	External distributed loads		Rigid link
	Vessel fixing		Custom equipment
	Structure		Non-circular component

[a)] for horizontal vessels
[b)] for vertical vessels
[c)] for column vessels

"View" and "Standard views" toolbars

The Table 3-2 describes the functions of the “View” and “Standard views” toolbar icons.

Table 3-2

Icon (name)		Function
	Front view	Full-screen model display in Z-Y plane (X-axis pointed away from screen).
	Back view	Full-screen model display in Z-Y plane (X-axis pointed toward screen).

Icon (name)	Function
Left-side view	Full-screen model display in X-Y plane (Z-axis pointed away from screen).
Right-side view	Full-screen model display in X-Y plane (Z-axis pointed toward screen).
Top view	Full-screen model display in Z-X plane (Y-axis pointed toward screen).
Bottom view	Full-screen model display in Z-X plane (Y-axis pointed away from screen).
Isometric view	Full-screen isometric model display.
Fit to the screen size	Full-screen model display in current view.
Zoom in	Zoom in using the left mouse button.
Zoom in/zoom out	Zoom in/out by moving the cursor up or down using the left mouse button.
Rotate around axis	Rotate model around axis by holding down the left mouse button and moving the cursor.
Rotate around selected point	Rotate model around selected point by holding down the left mouse button and moving the cursor.
Move	Move model using the left mouse button.
Cancel view	Return to previous view (before rotation, zoom, moving).
Repeat view	Repeat cancelled view change (rotation, zoom, moving).
Solid	Display model as a solid, 3D object.
Gradient	Display model in gradient (semitransparent) mode.
Beam	Display model as transparent beam.
Display filling	Display of calculated filling as translucent volume
Insulation and lining view	Display of created volumes of insulation and lining
Display service plaforms	Display or hide sites existing in the model
Colors by materials	Highlighting of components by color according to the “Materials using” panel
Perspective view	Display model in perspective view
Dimensions	Display dimensions of model components

Icon (name)		Function
	Elevation labels	Display elevation labels of shells, nozzles and platforms, stiffening rings
	Product level marks	Display product level marks by cavities
	Labels	Display model with component labels
	Origin point	Display origin point of solid model (relating to it, barycenters of components are calculated)
	Fine generation of model	The solid model is created in detail that can retard the work in case your computer is too slow.
	Accelerated model generation	Some components of solid model are created more simply.
	No model rebuild	Model rebuilding is blocked.
	Regenerate model	Forced rebuilding of the model with the elimination of artifacts left over from previous incomplete rebuilds
	Moving down of the component in the current branch (Ctrl+↓)	Changing position of the selected component in the model hierarchy relating to the components of the same level. This option is available only for daughter components (nozzles, rings, etc.).
	Moving up of the component in the current branch (Ctrl+↑)	

For quick move, zoom and rotation of the model, you can also use left, right and middle mouse buttons, respectively, while pressing the "*Ctrl*" key. To rotate the model around a selected point, you can use the right mouse buttonwhile pressing the "*Ctrl*" and "*Shift*" keys together.

When rotating the model around a selected point, X and Y coordinates (in the coordinate system of the screen) are determined by the mouse cursor position, while the Z coordinate ("depth") is determined by the current depth of non-transparent model component under the mouse cursor. If no non-transparent components are present under the mouse cursor, Z coordinate is set as equal to model's average depth.

3.11. Model tree

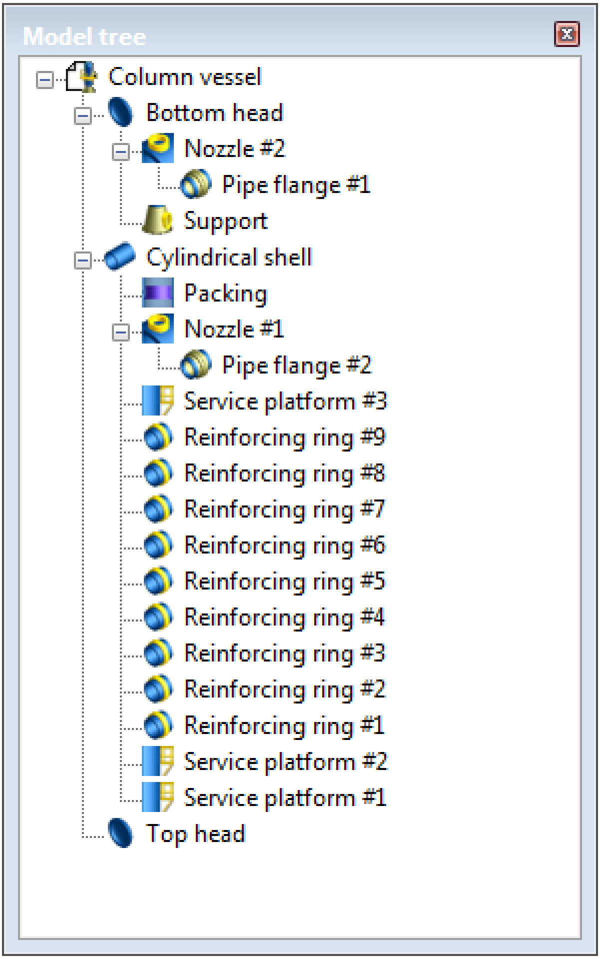
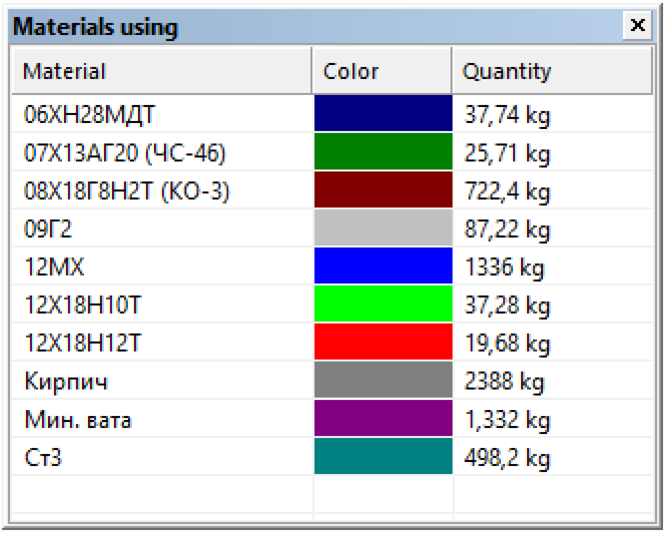


Fig. 3.11 "Model tree" Toolbar

A model tree is designed for a visual presentation of model structure and quick navigation. Elements of model are represented as scaled-down icons with names. The icons are interactive and have a pop-down menu. So, they provide easy access to component editing commands. A top-most component with model file name provides general data editing.

3.12. Materials using



Materials using

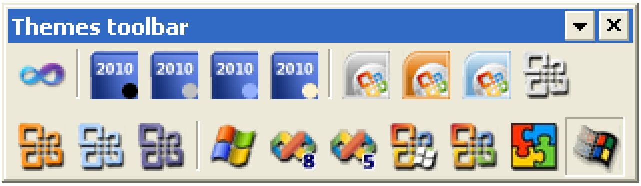
Material	Color	Quantity
06ХН28МДТ		37,74 kg
07Х13АГ20 (ЧС-46)		25,71 kg
08Х18Г8Н2Т (КО-3)		722,4 kg
09Г2		87,22 kg
12МХ		1336 kg
12Х18Н10Т		37,28 kg
12Х18Н12Т		19,68 kg
Кирпич		2388 kg
Мин. вата		1,332 kg
Ст3		498,2 kg

Fig. 3.12 “Materials using“ panel

This panel is designed for the rapid material consumption assessment. It displays a list of materials used in the construction, colors of materials used in  “Colors by materials” mode, and the estimate of the mass of each material in accordance with a given density. Mass is displayed in units that have been selected in the "Dimension" dialog. If the density of the material is unknown, or set zero, the item is considered to piece goods, and in the "Quantity" column shows the number of parts (eg, gaskets).

3.13. Themes toolbar, customization of toolbars and menus

Program has the ability to switch PASS/EQUIP interface style can be adjusted. To do this, choose the style in themes toolbar.



If the themes toolbar is hidden, you can enable it via **View→Toolbars →Themes toolbar**.

In addition, you can customize toolbars, add or remove toolbar buttons, and create new toolbars in **View→Customize**.

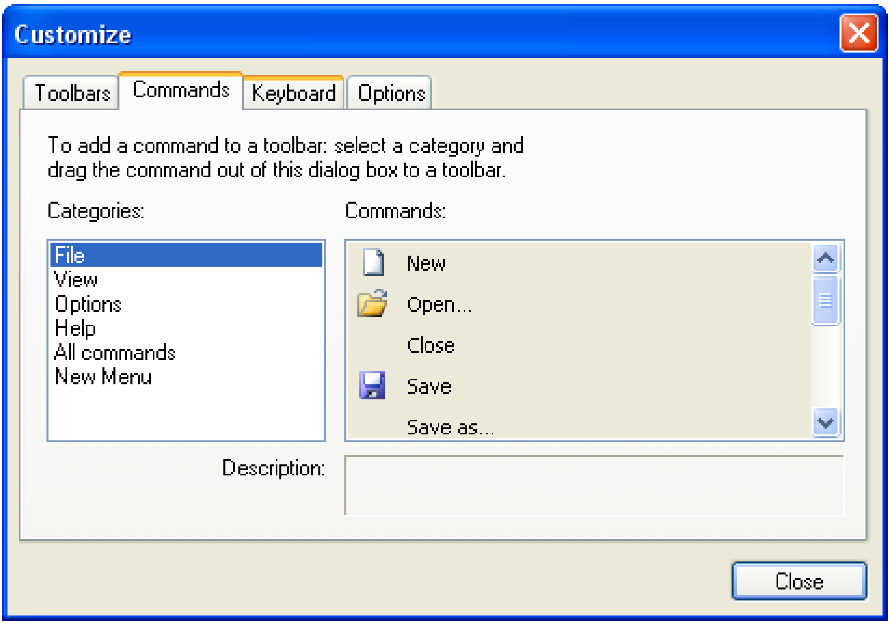


Fig. 3.13 Customization window

When customization window displayed, you can drag and drop. Desired toolbar buttons and menu commands with the mouse can be selected in the customization window by dragging and dropping.

3.14. Software settings

To show PASS/EQUIP settings, select "**Options**", then click submenu "**Settings**". The settings dialog box includes the following tabs and commands:

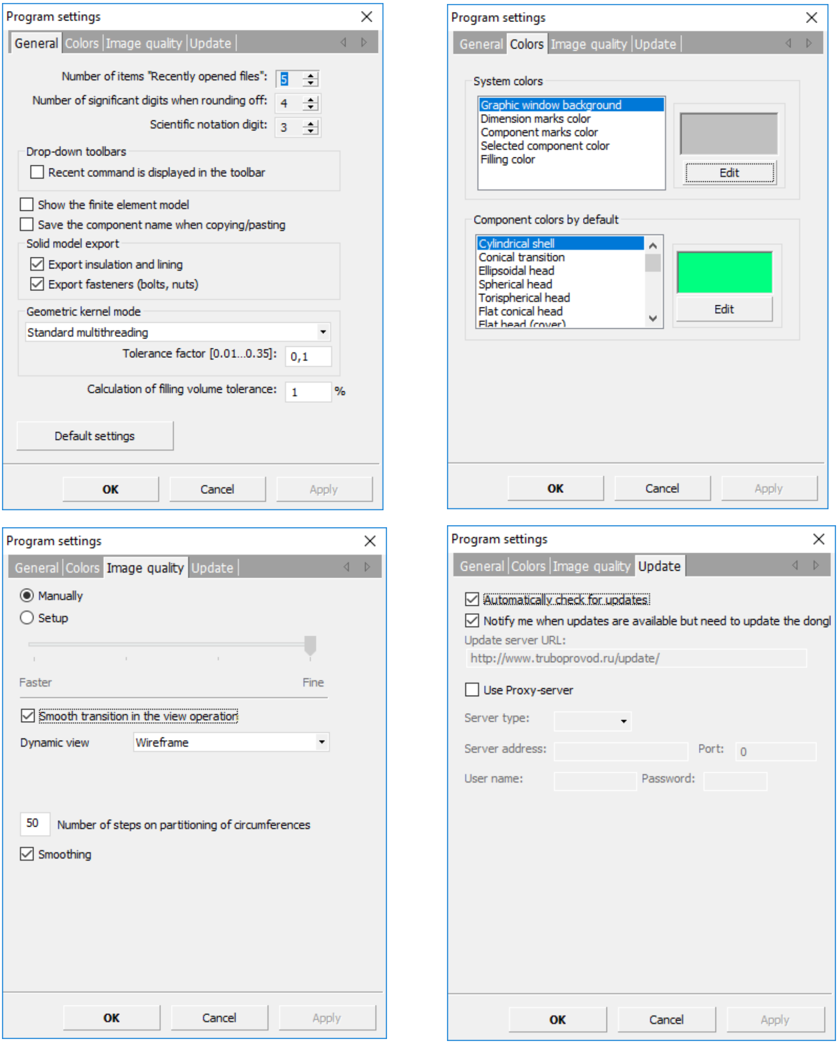


Fig. 3.14 Software settings

The Table 3-3 describes the options available within the tabs of the settings dialog.

Table 3-3

"General" tab	
Number of "Recently opened files"	Set the number of recent documents to be shown in the "File" menu.
Number of digits when rounding	Set the maximum number of digits when rounding numbers in output. For example: if this value is set at “3”, 1032.37 will be printed as 1030.
Minimum number of digits for scientific notation	Set the minimum number of digits when scientific notation should be used. For example: if this value is set at “4”, 10320 will be printed as $1.032 \cdot 10^4$, while 1270 will be printed as is (depending on rounding; see above). Note: for numbers with notation of less than 10^3 and greater than 10^{-1}, this setting is ignored.
Recent command is displayed in the toolbar	Set recently used command to add model component as default command in the toolbar.
Show finite element model	While working with PASS/EQUIP, the model can be shown as a finite element beam model. This model provides better control over software functions, but can slow down software operation.
Show model fill	Calculated model fill will be indicated by colored dots
Filling calculation (Faster - Fine)	PASS/EQUIP program implements calculation of vessel filling based on a statistical method of calculation of the volume integral, when the model is filled with randomly generated points (Monte Carlo method).This setting controls the number of generated points.The more points, the better the result, but the calculation is longer (in the fastest mode, the accuracy is ~5%).
Export insulation and lining	Establishes, whether there is a need to include insulation and lining, when exporting solid

	model. If lining is set as a plating (Fig. 3.27), it will be always exported.
Export fasteners (bolts, nuts)	Establishes, whether there is a need to include fasteners (bolts, nuts, studs, etc.), when exporting solid model
Geometric kernel mode	Includes multi-threaded mode of operation of some functions of geometric kernel. Working with the model in multi-threaded mode is faster, but in some (rare) cases, the program may fall.
Accuracy control parameter	A numerical value that controls the accuracy of the mathematical functions of the geometric kernel in the construction and calculation of mass-dimensional characteristics. The higher the value, the faster and more accurately the calculation is performed
Accuracy of calculation of filling volume	A value of the relative error of calculation of the filling, when selecting option "Calculation by the specified volume of product." For example, at a value of 1% and specified target volume of 1000 liters, the calculation will be considered successful, if it gives any value in the range of 990 to 1010 liters. A small relative error can extend the calculation for a complex configuration of cavity of the vessel (heat exchangers, etc.)
Materials DB	Allows you to configure the path to the database of user materials located in the centralized access (for example, on a network drive), or use a local database (by default)
"Colors" tab	
System colors	Customizes colors of view window elements
Element colors	Customizes default colors of model components. New model components will be created with this setting. To apply changes to components that were previously created, go to "**Components**"→"**Change color**"→"**Default**

	colors” or use the appropriate icon .
"Image quality" tab	
Manual setup	Set image options manually.
Default	Use default values.
Smooth transition in view operations	If selected, changing to selected view (standard views and "Show window" function) is performed smoothly.
Dynamic view	Choose view during dynamic operations: moving, zooming and rotation: "Normal" – aligns with current view; "Beam" – beam model. May significant accelerate view operations on slow PCs. "3D box" –3D box circumferencing the model. Used on slow PCs.
Number of sections for circumferences	Set accuracy for displaying curvilinear surfaces (cones, spheres) by setting the number of sections.
Anti-aliasing	If selected, a full-screen image anti-aliasing is applied to correct imperfections. Requires a high-performance OpenGL video adapter.
"Update" tab	
Check for updates at each run	Automatically check for updates at the application run
Notify of updates, which require dongle updating	Notify of updates, which can be installed after dongle updating.
Use Proxy-server	Connect with updating server via proxy server (required, if Internet is launched via HTTP through the proxy server)

3.15. Software update system

Regular program updates provide user with the latest version of the software.

An update system can check for updates in automatic or manual mode, as well as download and install updates on PC.

For the proper work of the system a dongle is required. If the dongle is not available, check and installation of updates cannot be performed.

Check for updates is performed automatically at the program launch, or manually from **Help→Check for updates** menu**.** Automatic check can be switched off in the parameter settings window, tab **“Update”,** check button **“Check for updates at each run”.**

Check and installation of updates is performed up to maximum allowed version number, which is specified in the dongle. Software update system can notify a user of the available updates to the latest versions, which are newer than those allowed by the dongle, if the following item is switched on: **”Notify of updates, which require dongle updating” in the parameter settings window, tab “Update”.** If this message appears, such updates become available after the dongle update to the required version (see i. 2.4).

To install updates, system administrator’s rights are required. There can be a request of UAC (Windows User Access Control system) about permission of install.exe launch during installation process. . For proper updating, install.exe shall be launched with administrator’s rights.

Proxy settings may be required for connecting to update server. The same proxy settings as in a web-browser should be set. In the case of Internet Explorer these settings can be found in the tab **Connections→Network properties** of the Internet Options. When being installed, the program offers to use as default settings for these options, installed in the system.

3.16. Measurement unit settings

Before creating a model (and at any other time), you can set dimension and load measurement units. Element properties are recorded using internal units by PASS/EQUIP and automatically re-calculated using these settings. The units selected in these settings are displayed in the output.

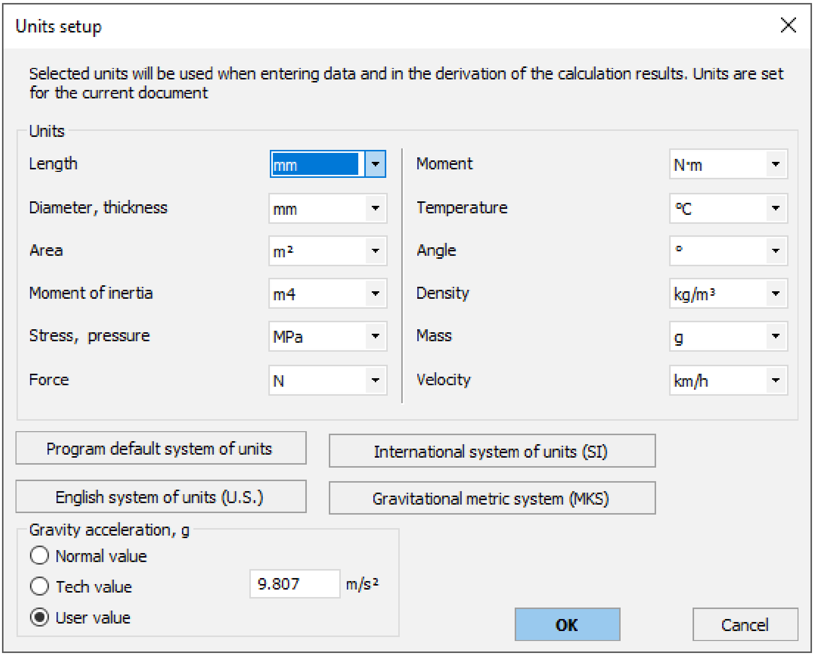


Fig. 3.15 Units settings

Measurement units need not follow the same measurement system (eg, diameter in mm and moment in N·m). Therefore, when viewing equations in output, the final result may not coincide with intermediate calculations, which does not constitute an error (M = 2 [N] * 1000 [mm] = 2 [N·m]). To avoid this, set units from the same measurement system (for example, all linear dimensions are in mm and moment in H·mm).

Buttons with systems of units are useful for quick assignment of the whole complex of dimensions used in the corresponding system.

The “Gravity acceleration” option allows more flexible adjustment of the weight, seismic and inertial loads calculation.

3.17. Data input

You can start building the model from a cylindrical shell or conical transition or from any head. First component is placed from zero position along Z-axis: from left to right for horizontal vessels and from the bottom up for vertical vessels. The following components are joined or inserted to existing model components.

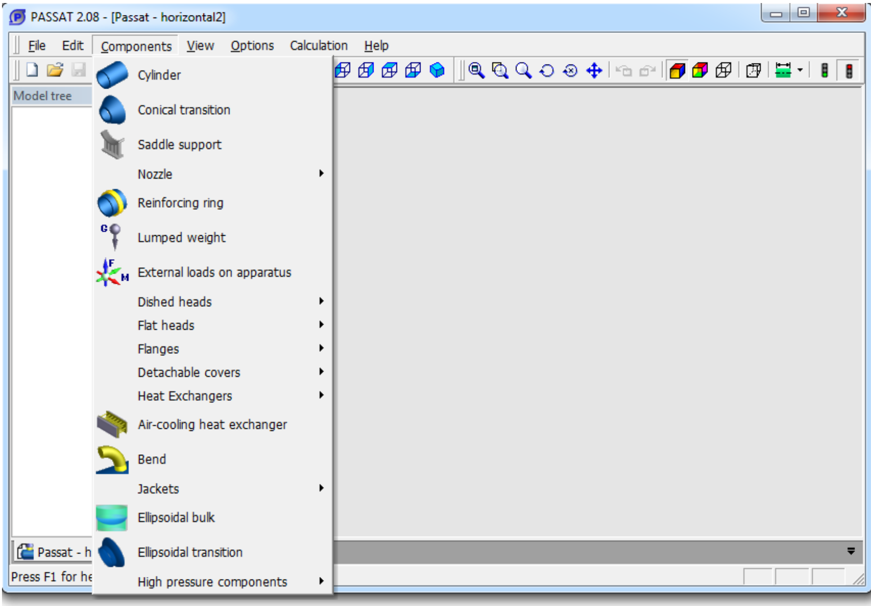


Fig. 3.16 "Components" menu

After you select **"Components" Add ▸** (or press the corresponding icon in the right column of the screen), possible joint or insertion points are determined and you will be asked to select the desired joint or insertion location:

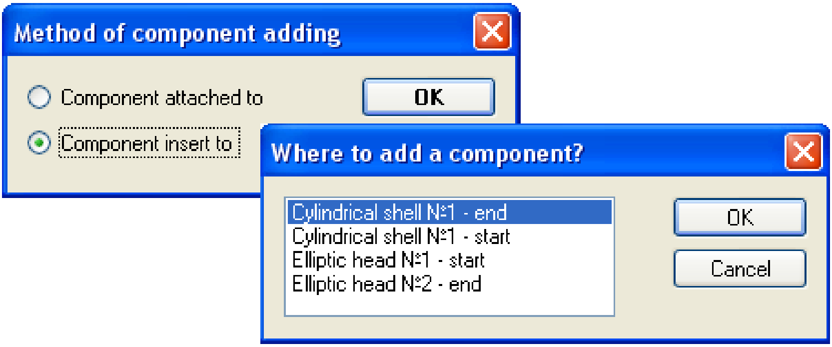


Fig. 3.17 Model building

After the new component is placed, a dialog with all required dimensions, materials and load properties data will be displayed. To input data, press “Enter” in the desired field. Element properties will then be recalculated if necessary.

Some options are common to different components and work in a similar way.

3.17.1.1 Component name

Set component name ("Cylindrical shell No…" is used by default). This name will be used during editing, deleting and assigning of adjoined components, and displaying output.

3.17.1.2 Code

Set the standard to be used for analyzing this component. When you change the selected regulatory documents, input data indications are changed as well, and their values are automatically recalculated if necessary. The code (normative document) can be assigned individually for each component, i.e. GOST 34233.2-2017 for shell and ASME VIII-1 for nozzle. At that, if during calculation the nozzle requires the shell parameters calculated as per ASME, they will be calculated, despite the shell code specified.

3.17.1.3 Temperature

Temperature, at which material properties in operation state are calculated.

3.17.1.4 Design pressure

Internal or external pressure in operation state. Does not include hydrostatic pressure of liquid, if present. Hydrostatic pressure for each component is calculated individually based on fill estimation. Excessive pressure with allowance for hydrostatics is calculated using the following rule (for external pressure p is negative):

$p' = \pm p + \rho \cdot g \cdot h$, if p is internal, or if p is external but $|p| < |\pm p + \rho \cdot g \cdot h|$;

$p' = \pm p$, if p is external and $|\pm p| \geq |\pm p + \rho \cdot g \cdot h|$.

3.17.1.5 Design values calculation

To calculate thickness, allowable pressure and other properties without existing the component properties window, use the **"Calculate values"** button. These parameters are defined more simply and approximately, without consideration for filling of the vessel, influence of the neighboring components, etc. The final result of calculation may differ from this value.

3.17.1.6 Material

Material selected from database (GOST 34233.1-2017, PNAE G-7-002-86, GOST R 54522-2011, ASME II Part D, EN etc.), or is set by user, where required properties at operating, test and assembling temperature (20^0C) must be set.

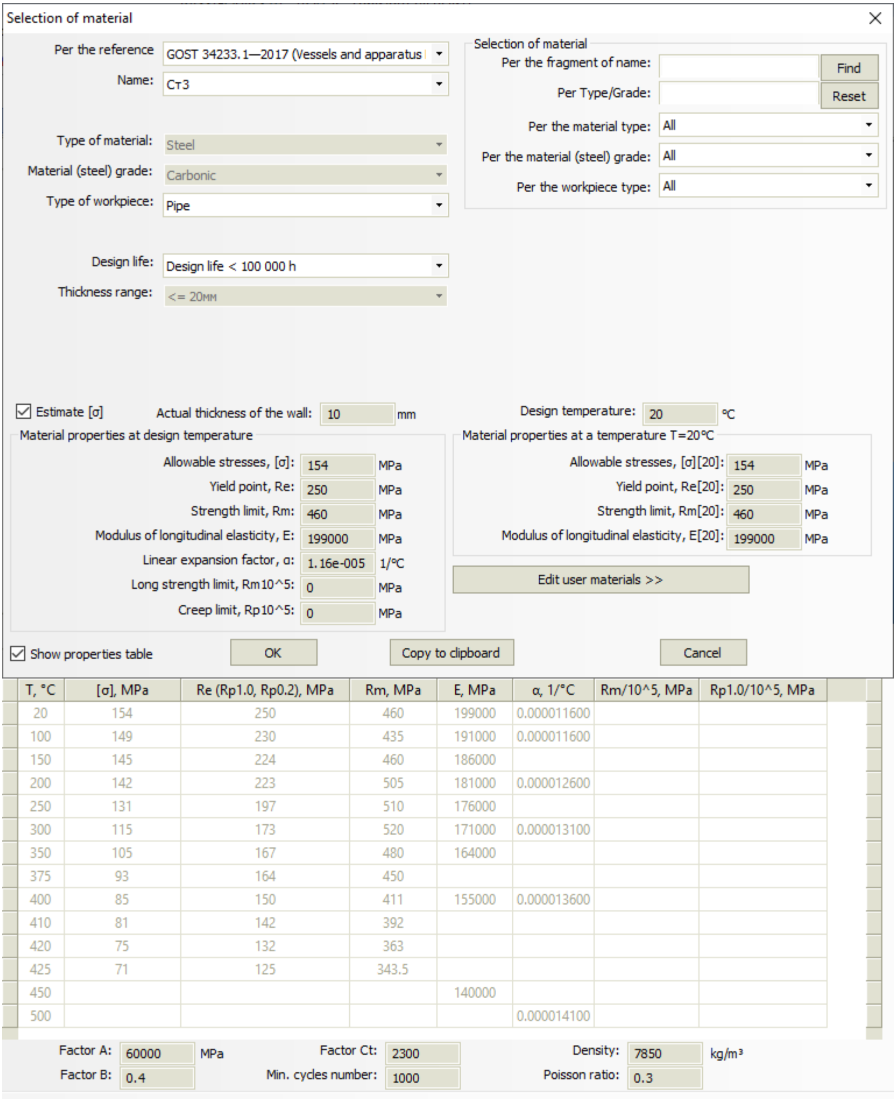


Fig. 3.18 Standard material properties

Material database consists of two parts: group of standard materials and group of user’s materials. First group cannot be changed by user. To edit the second group, “Edit user’s materials” shall be selected.

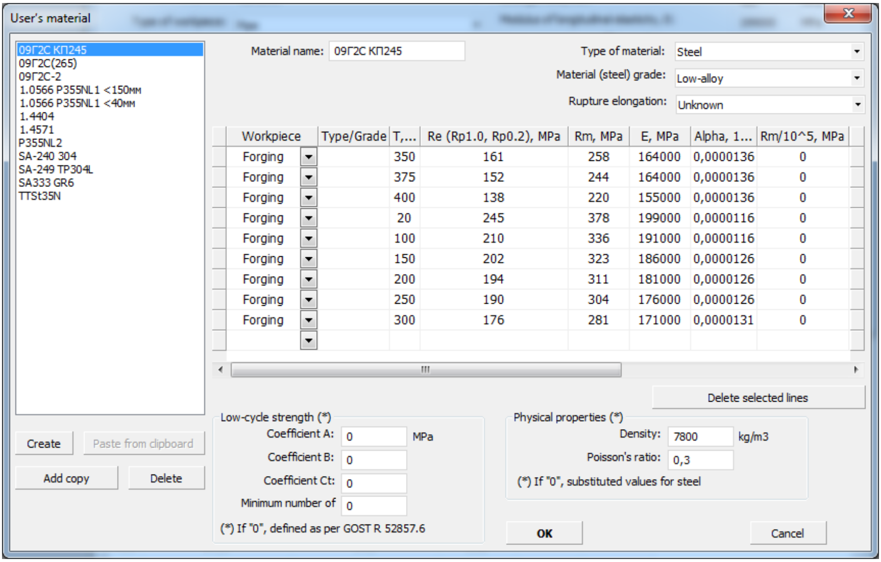


Fig. 3.19 User defined material properties

In the user's materials editor the following operations are available:

- Adding new "empty" material ("Create");
- Copying table of properties of any of the existing materials as a whole (the command "Copy to clipboard" in the table), and the subsequent insertion to the user material (command "Paste form clipboard");
- Deletion of material "highlighted" in the list ("Delete");
- Creation of the new material and copying into it the properties of the material, which is "highlighted" by the cursor in the list ("Add copy");
- Deletion of the group of lines with material properties ("Delete selected lines");
- Material renaming (the name of material is edited in the same field);
- Setting of material properties, depending on temperature, thickness, workpiece type, parameters Type/Grade and Class/Condition/Temper (by analogy with ASME II Part D).

After pressing "OK" button a new material will appear in the group "User's materials".

Material properties at given temperature are determined by linear interpolation. Data can be set randomly, depending on the temperature — the program will sort them automatically, in the order of temperature increasing. In addition to steel, cast iron, nonferrous metals and titanium alloys can be entered. A and B material properties and correction factor C_t are set for steel, non-ferrous metals, cast iron and titanium alloys when calculating low-cycle fatigue.

If some value is initially equal to 0, then at calculation in will be set automatically, as per GOST 34233.6-2017, based on the material type and grade.

When you add a new material, its properties are saved both in the database and in the model file. When transferring data file to another PC, PASS/EQUIP will read material data and add it to local database if necessary. If a material with the same name already exists in database, properties from the database and not the model file will be used.

In case of simultaneous access of several users to the publicly available database (for example, when it is placed on a network drive), only the first user can edit (for other users, a message appears about the impossibility of editing at the moment).

3.17.1.7 Standard dimensions

By this command you can select a component from standard items database via clarifying filters (from more general parameters to the particular ones)

3.17.1.8 Negative tolerance

By >> key you can select this value from database as per different standards. A user can select only those variants, which correspond to the defined nominal thickness of the wall.

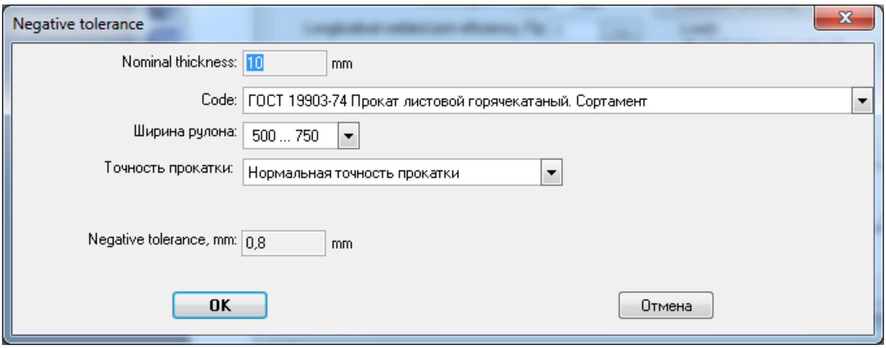


Fig. 3.20 Negative tolerance

3.17.1.9 Weld strength ratio

This value is set based on weld type and materials used via >> button.

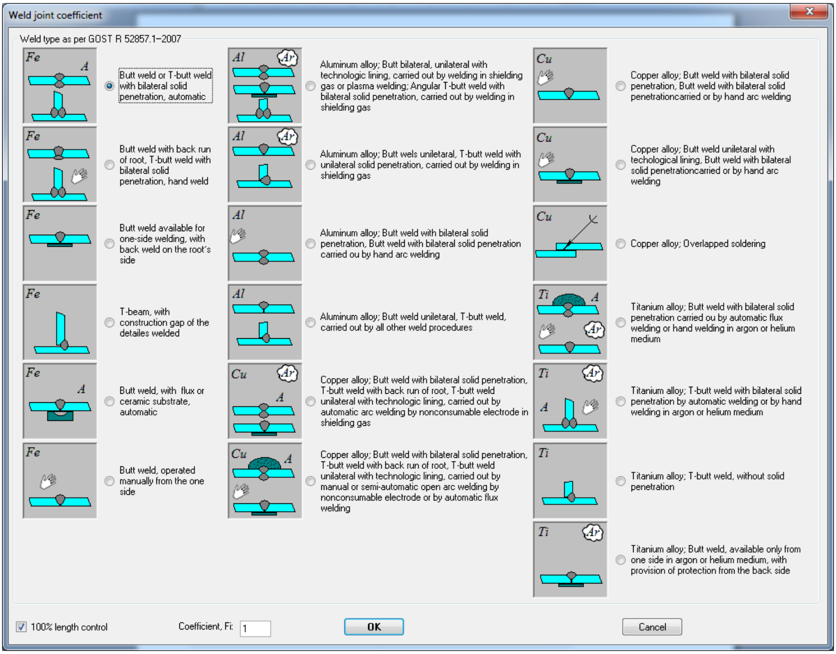


Fig. 3.21 Weld strength as per GOST

Selection of this parameter depends on the accepted calculation code. Ff calculation is selected as per ASME VIII-1, weld strength factors are assigned concurrently (Fig. 3.22).

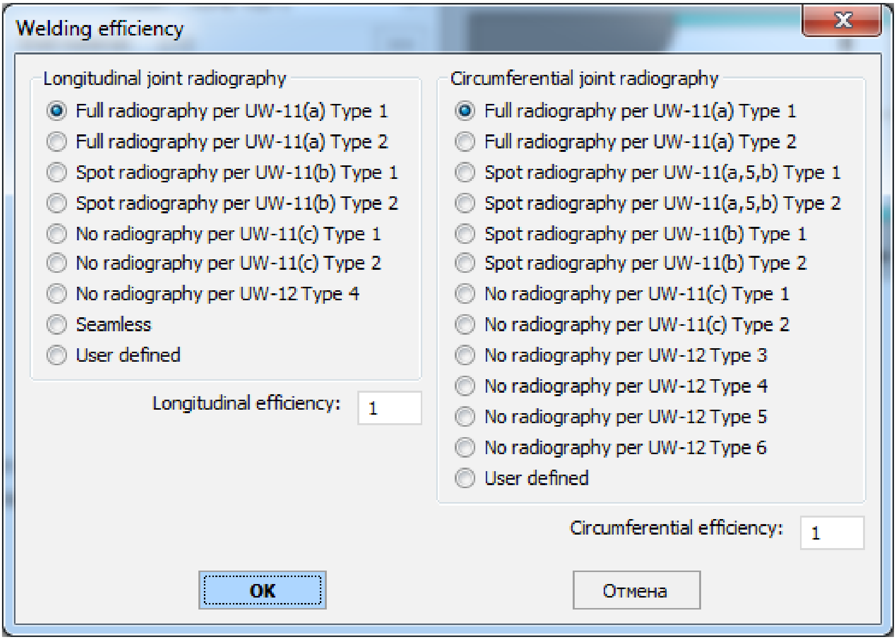


Fig. 3.22 Weld strength as per ASME

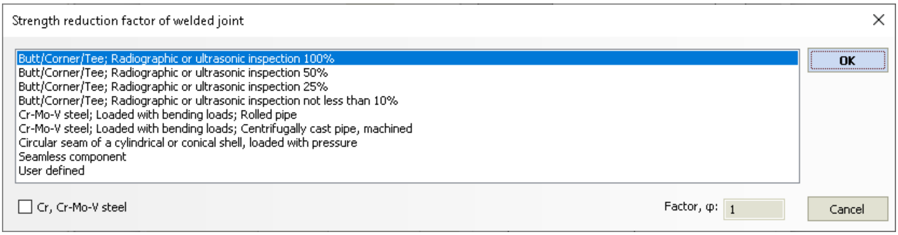


Fig. 3.23 Weld strength as per PNAE G-7-002-86

3.17.1.10 Insulation and lining

In the presence of insulation, for automatic weight accounting, you should specify its thickness, as well as density or mass (for complex heterogeneous insulation). When assigning insulation, the program takes into account the change in the outside dimension of the component, when calculating wind loads.

Thickness and density of insulation can be selected from database according to the current regulations (Fig. 3.26, Fig. 3.27).

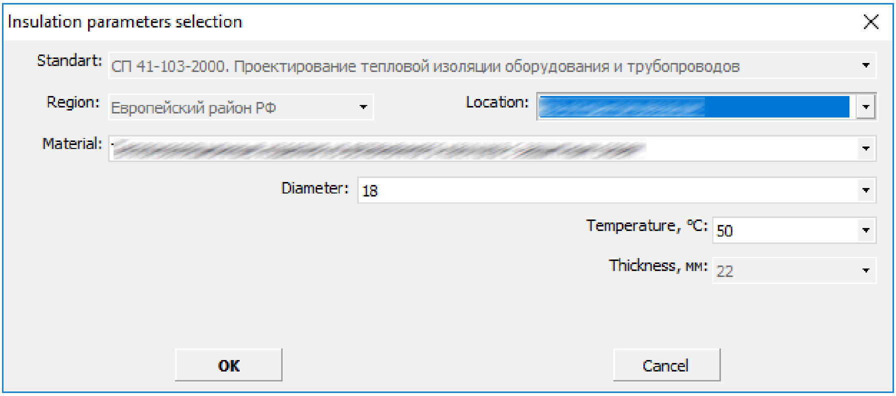


Fig. 3.24. Thermal insulation thickness selection

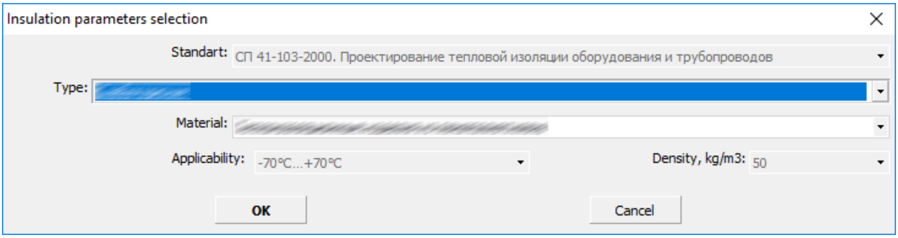


Fig. 3.25 Thermal insulation density selection

By selecting "Calculate" option, you can receive a lot of components of thermal insulation via the calculation module of the **"Insulation"** program. Input data for calculation are the geometric dimensions of the model component, its temperature, climatic parameters and project data specified in the dialog "Data for calculation of insulation" (i.3.9)

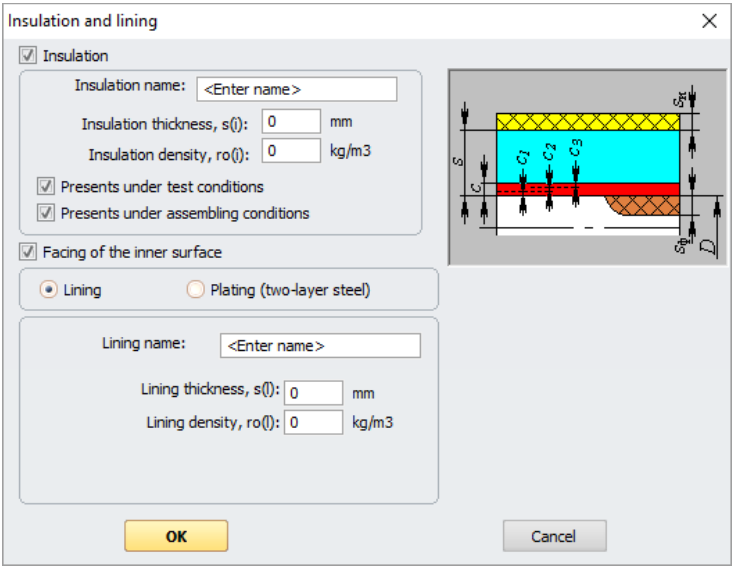


Fig. 3.26 Non-metallic insulation and lining

Option "Presents in test/assembly conditions” influences on weight of the component and its outside ("wind") diameter in appropriate conditions.

In the presence of lining, for automatic weight accounting, you should specify its thickness and density.

For some components (shell, head), it is possible to take lining as plating (double-layer steel wall, Fig. 3.27). At that, the calculation takes into account the wall thickening and changes in allowable stresses.

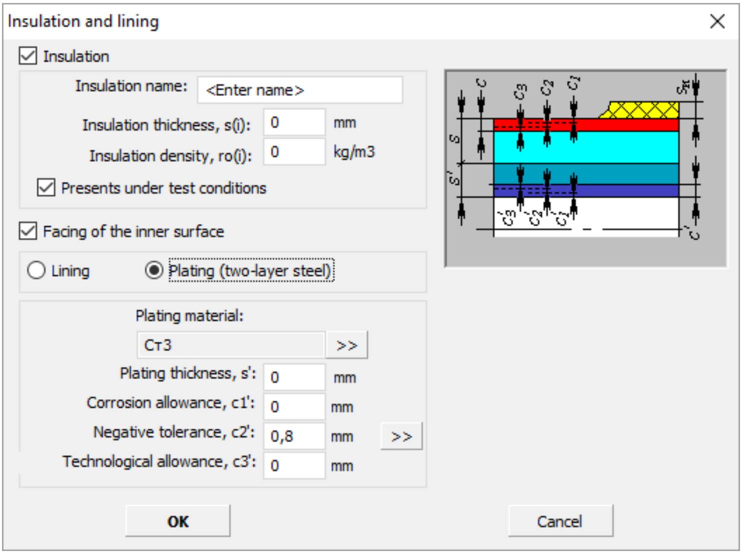


Fig. 3.27 Insulation and lining suitable for plating

For tube sheets of heat exchangers, two-sided cladding with recalculation of the allowable stresses of the carrying layer is provided (Fig. 3.29).

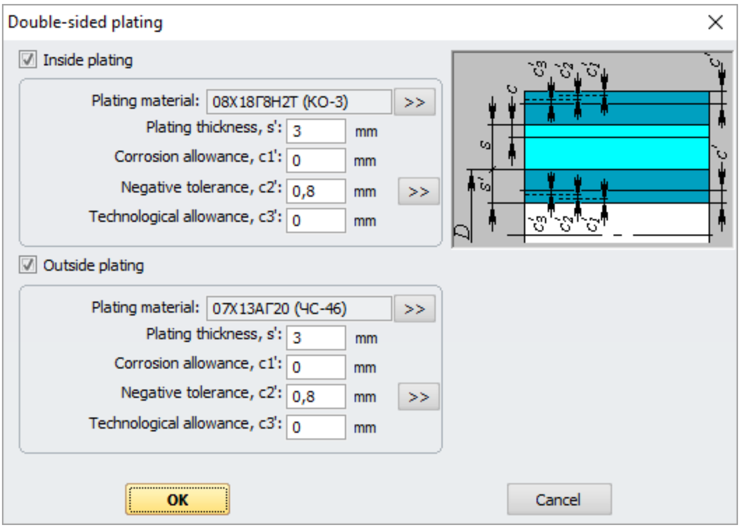


Fig. 3.28 Two-sided cladding

3.17.1.11 Low-cycle fatigue

For low-cycle fatigue analysis, load properties and weld type must be set, depending on adjoining nodes or vessel components.

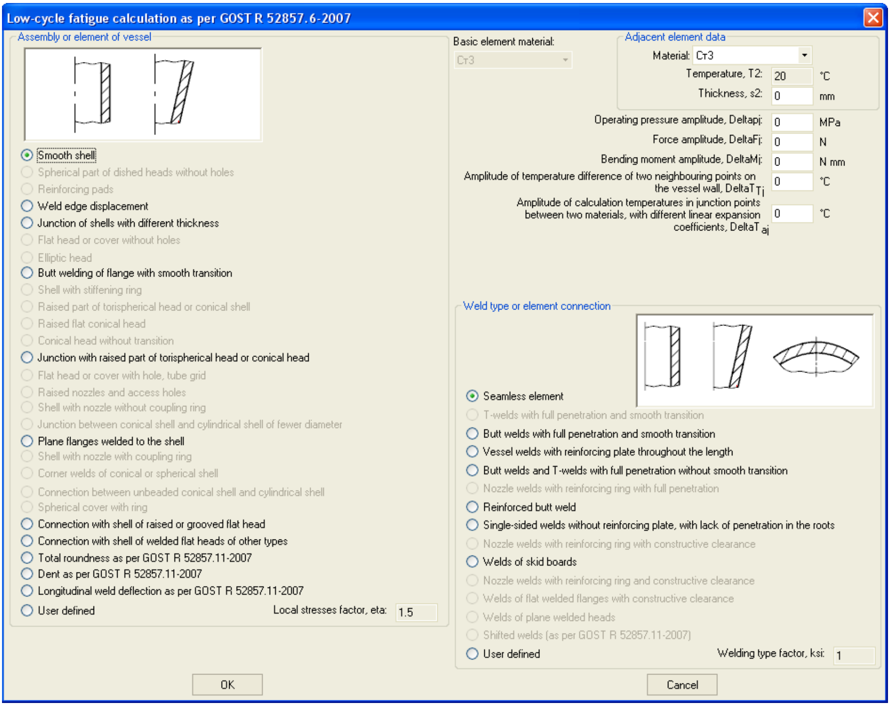


Fig. 3.29 Local stress factors

3.17.1.12 Defects according to GOST 34233.11-2017

If any defects are found, an additional analysis will be performed. Defect type and properties can be set via 

button.

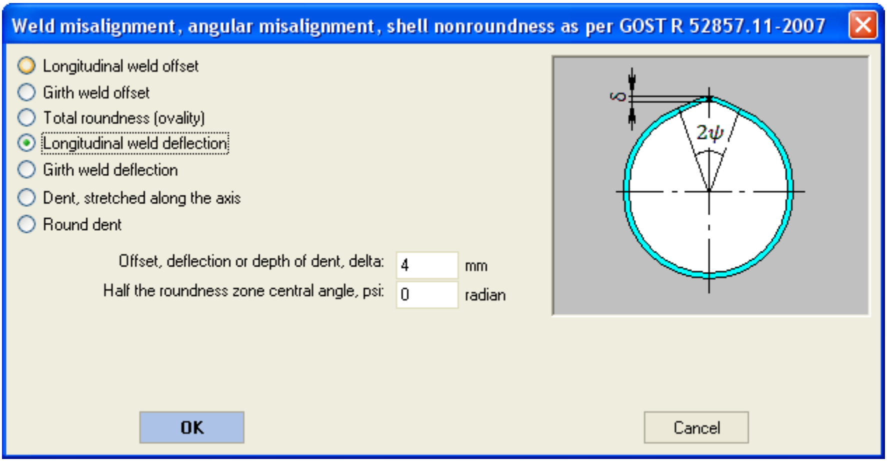


Fig. 3.30 Defects as per GOST 34233.11

3.17.1.13 Space in the component

In the simplest case, the vessel has one internal volume, and content properties are specified in the general data (Fig. 3.8). However, in some cases, the vessel has two or more isolated volumes (jackets, heat exchangers, vessels with separating walls). In this case, it is necessary to set parameters of filling of the subsidiary volume.

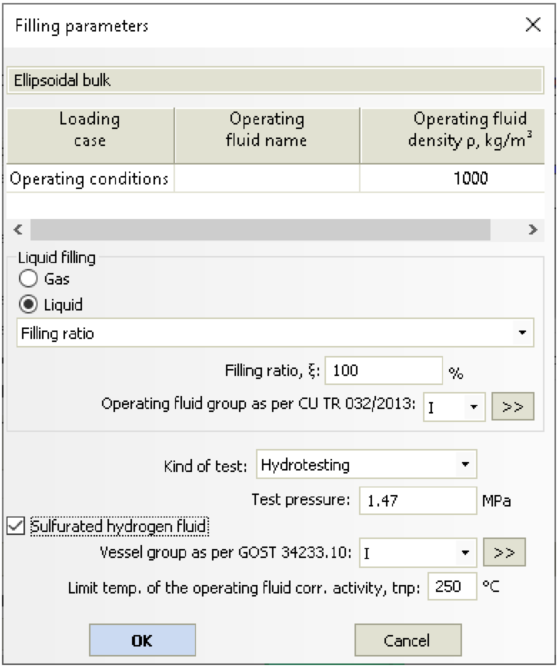


Fig. 3.31 Space in the component

This dialog works similar to the general data dialog (Fig. 3.8), but it is applicable only to subsidiary volume properties.

3.17.2. Cylindrical shell

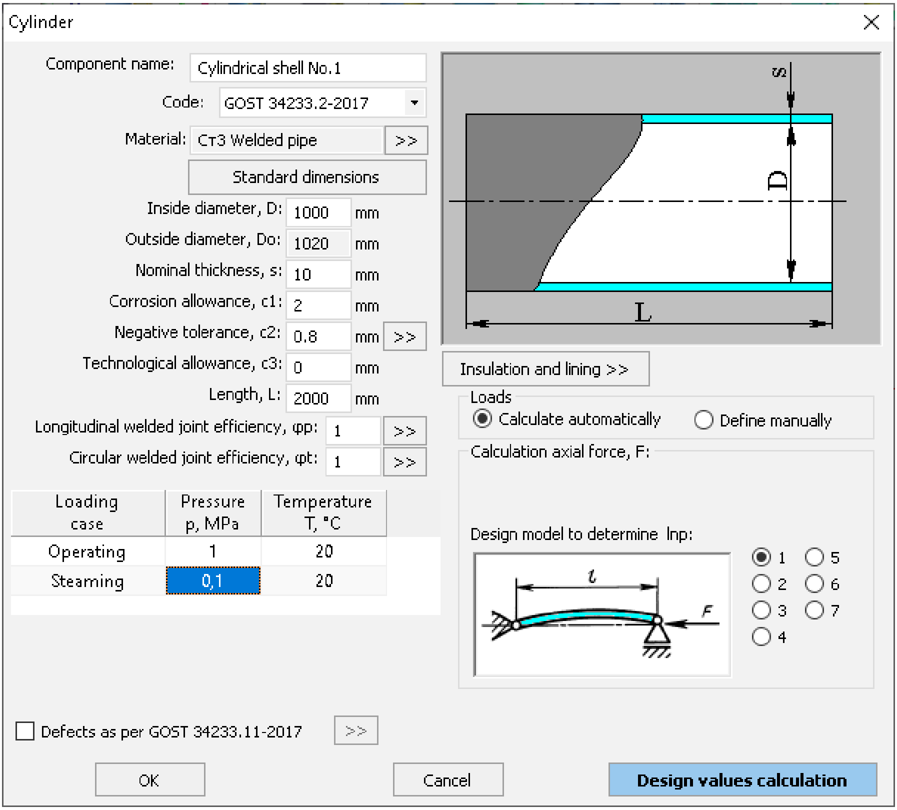


Fig. 3.32 Cylindrical shell

Code - Set the standard to be used for analyzing this component. Calculation as per GOST 14249-89, GOST 34233.2-2017, EN13445-3, ASME VIII-1 is available.

Standard dimensions – with this command one can select a number of preferable shell sizes (or reject it), indicate the intended type of workpiece (flat steel or pipe), and specify the standard of the workpiece. At that, wall diameter and thickness will be set automatically, as well as a negative allowance.

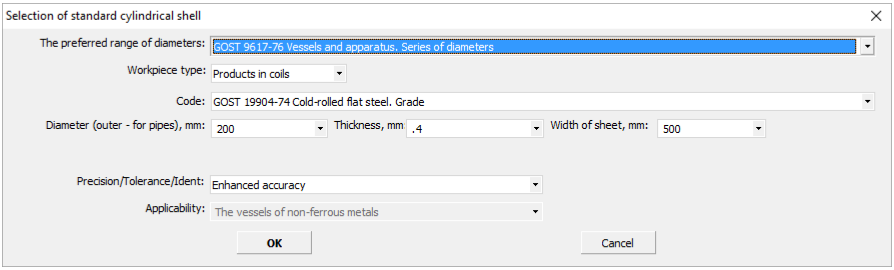
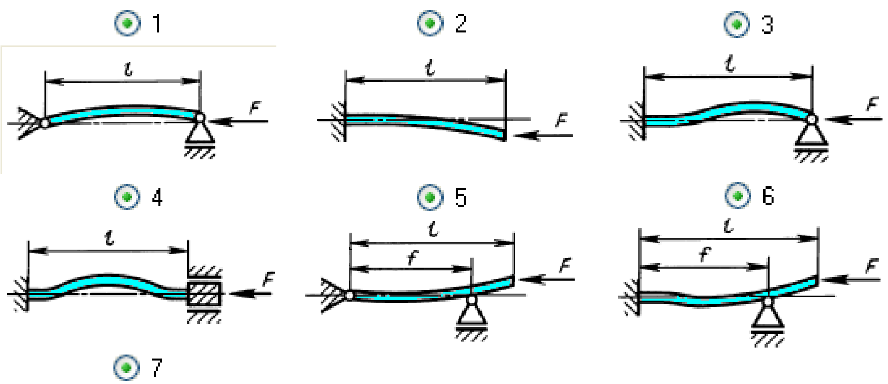


Fig. 3.33 Standard shell selection from database

Loads – if “Define manually” is selected, external loads and how they influence the component must be defined (see below). Input loads are considered only when analyzing this component and **are not transferred** to supports, adjoining components, etc. If “Calculate automatically” is selected, maximum loads are determined automatically based on restraint properties and properties of loads on all model components, weight loads from material and component content, etc.

Bending moment, intersecting force, axial force, design model are determined based on a preliminary analysis of external forces and moments affecting the shell.

In the presence of compression forces on the shell, design model is determined according to GOST 34233.2–2017 (GOST 14249–89), as shown on Fig. 3.34. Lengths of external pressure and axial force are calculated automatically based on the structure of the model as a whole.



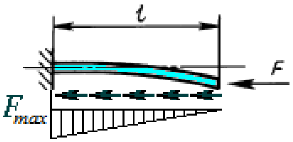


Fig. 3.34 Design models for determining the overall shell stability

At calculation of loads using FEM (finite element method), cylindrical shell is modelled by a chain of beam elements of ring cross-section with weightless nodes (Fig. 3.35). Uniformly distributed lengthwise load is applied to each chain element.

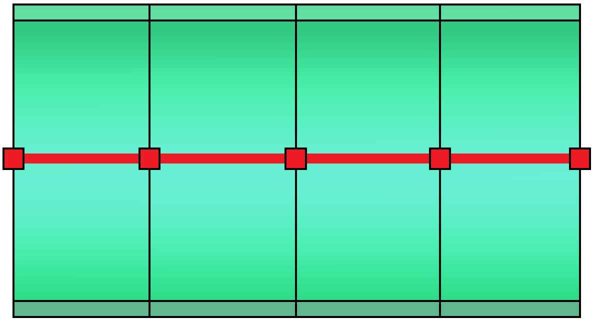


Fig. 3.35 Modeling a cylindrical shell with beam elements

3.17.3. Conical transition

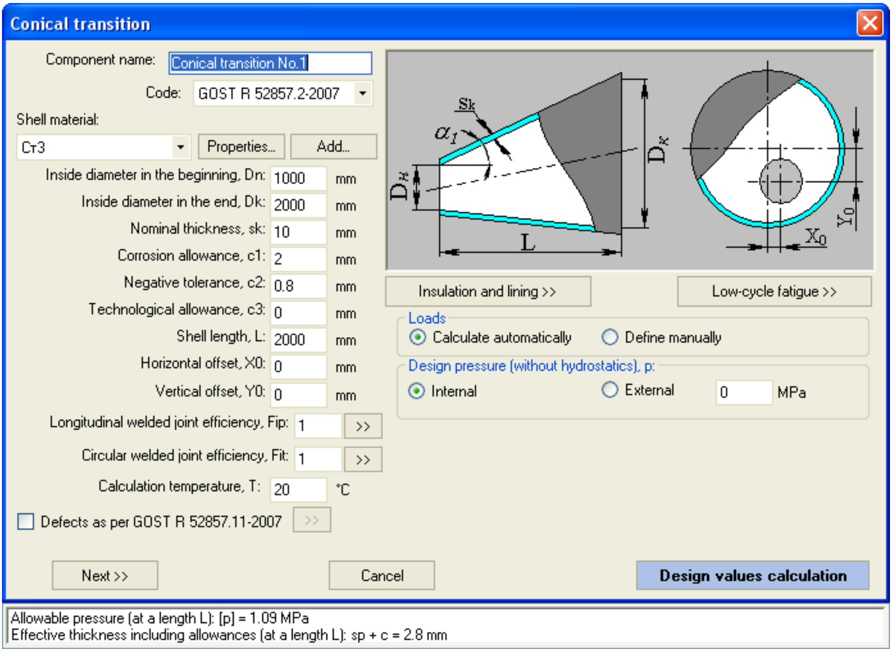


Fig. 3.36 Conical transition

Component name, code of standards, material, dimensions, weld strength factors, insulation and lining, and load properties of conical transitions are set in the same way as those for cylindrical shells.

Horizontal and vertical displacement is calculated for eccentric conical transitions. Use the Next >> button to switch to a list of joints with neighboring components. Joint structure is according to GOST 34233.2-2017 (GOST 14249-89).

At calculation of loads using FEM method, conical transition is modelled by a graduated chain of beam elements of constant ring cross-section (Fig. 3.37). To each chain element a uniformly distributed lengthwise load is applied, amount of which depends on the mean cross-section diameter at the given area.

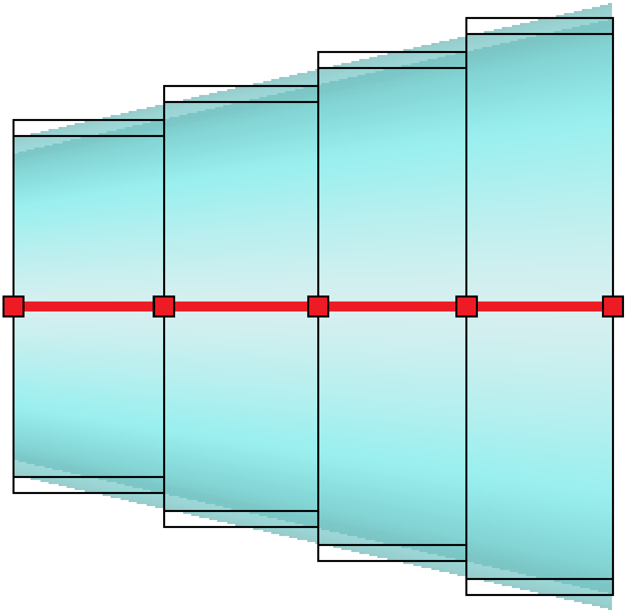


Fig. 3.37 Modeling a conical transition with beam elements

Reinforcement elements in the transition ends are modelled similarly to the cylindrical shell.

3.17.4. Dished head

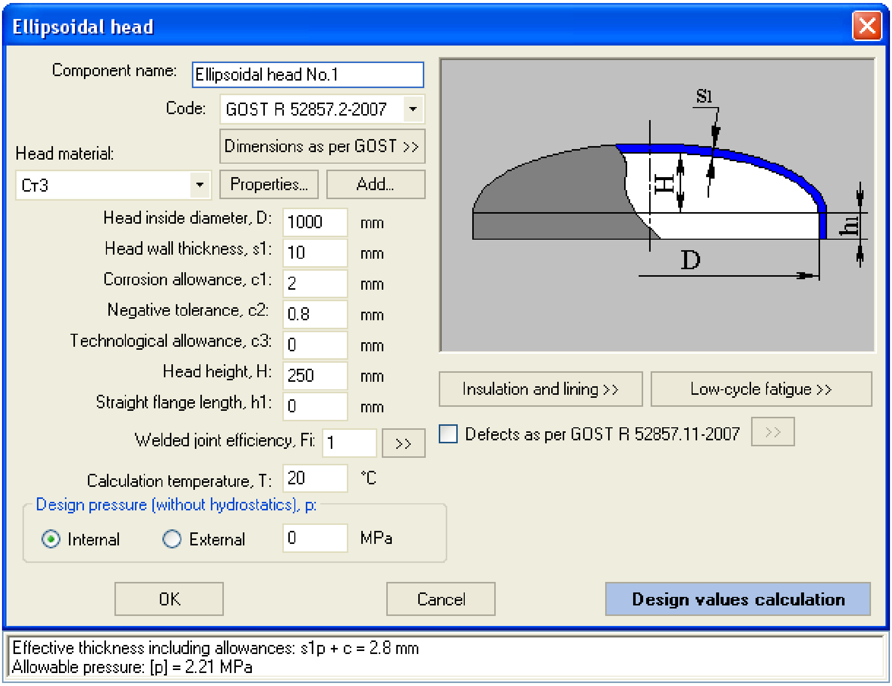


Fig. 3.38 Ellipsoidal head

Component name, code of standards, dimensions according to GOST, material, dimensions, weld strength factors, insulation and lining, and load properties for dished heads are set in the same way as those for cylindrical shells.

At calculation of loads using FEM method, dished heads are represented as a pair of weightless beam elements with node in the point, which corresponds to the head gravity centre (Fig. 3.39).Cross-section of elements is considered to be constant and corresponding to the cross-section of the head foundation. Head weight is considered to be lumped and is applied to the centre of gravity (yellow node).

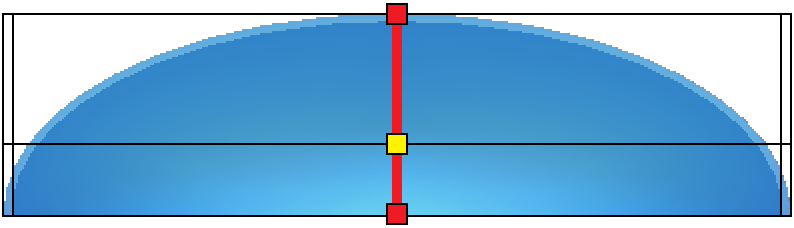


Fig. 3.39 Modeling a dished head with beam elements

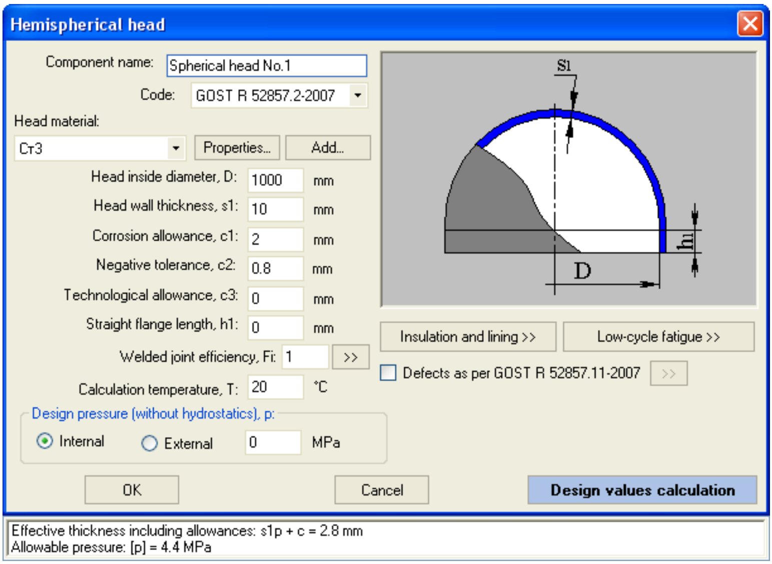


Fig. 3.40 Hemispherical head

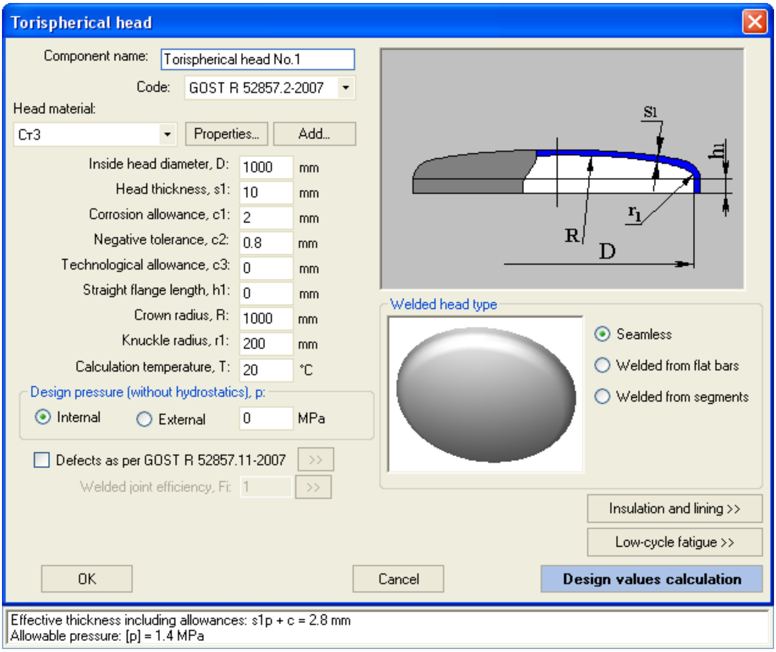
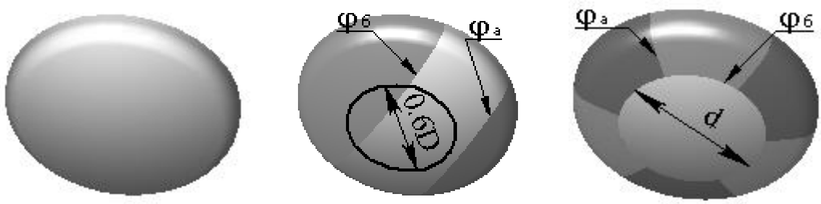


Fig. 3.41 Torispherical Head

Torispherical head type is determined according to GOST 34233.2-2017 and can be seamless pressed, welded from flat bars or welded from segments (Fig. 3.42). Weld strength factors must be set welded heads.



Seamless Welded from flat bars Welded from segments

Fig. 3.42 Torispherical head types

3.17.5. Flat conical head

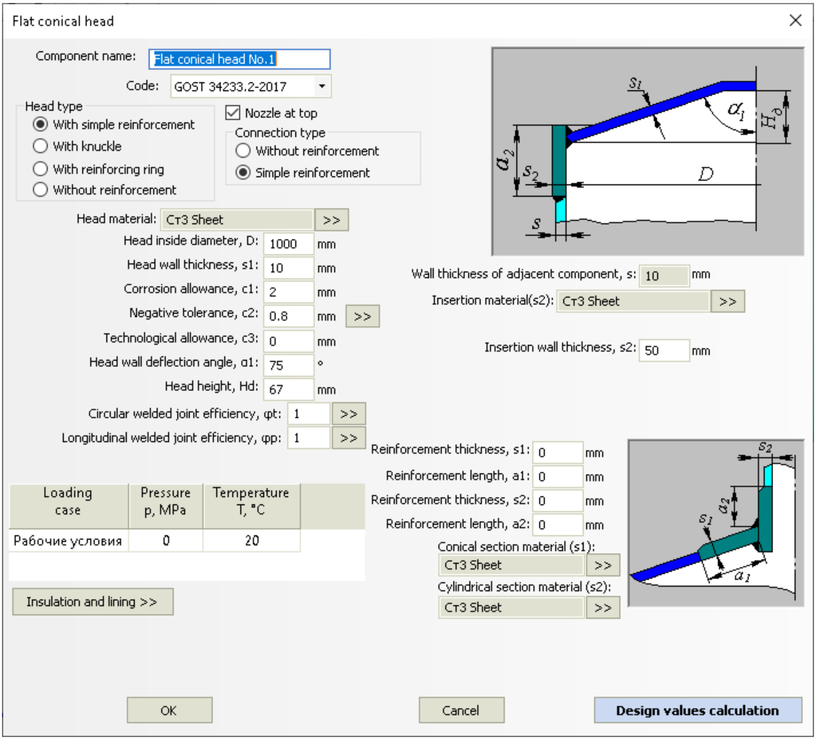


Fig. 3.43 Flat conical head

Component name, code of standards, material, dimensions, weld strength factors, insulation and lining, and load properties for flat conical heads are set in the same way as those for cylindrical shells. Head type is determined according to GOST 34233.2-2017.

“Nozzle at top” option is used in cases when the modeling of the vessel with a conical transition gives an incorrect result (for example, for horizontal vessels on saddle supports).

3.17.6. Steep conical head

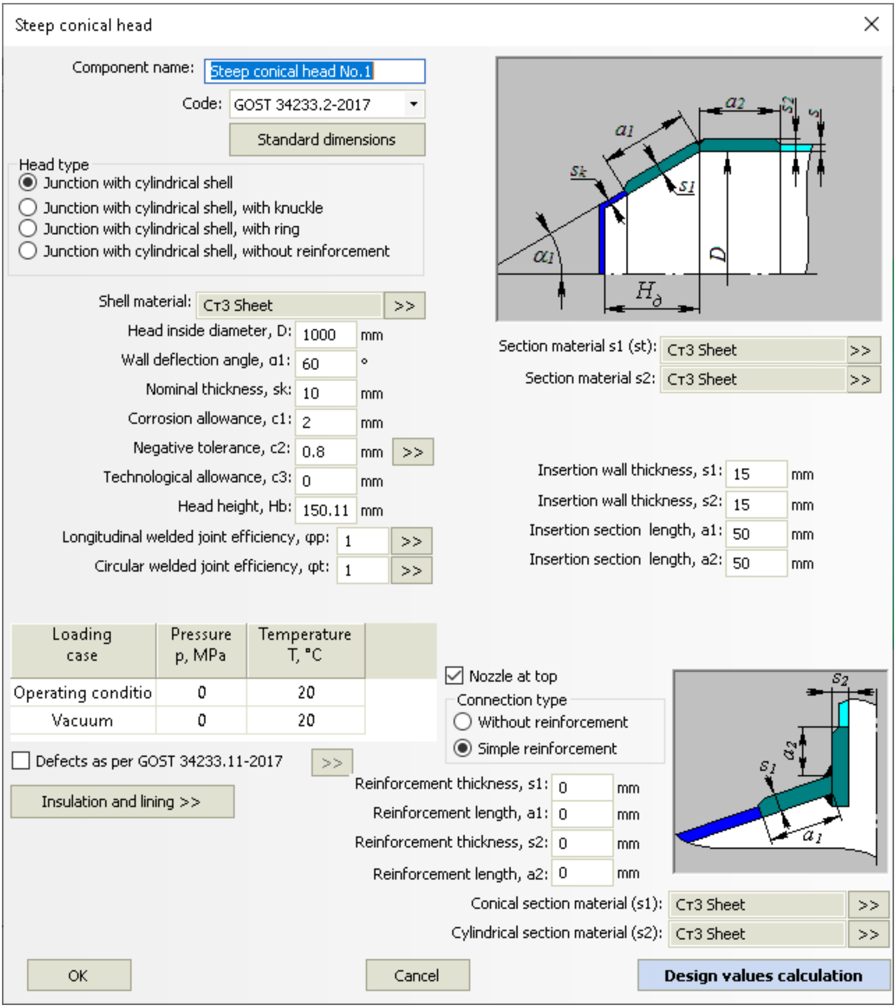


Fig. 3.44 Steep conical head

Component name, code of standards, material, dimensions, weld strength factors, insulation and lining, and load properties for steep conical heads are set in the same way as those for conical transitions.

"Nozzle at top" option is used in cases when the modeling of the vessel with a conical transition gives an incorrect result (for example, for horizontal vessels on saddle supports).

3.17.7. Flat head

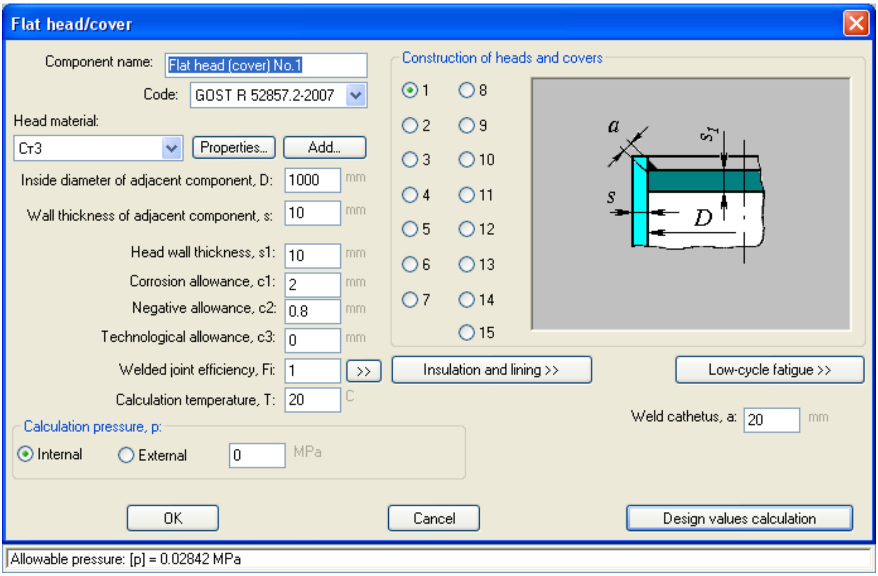
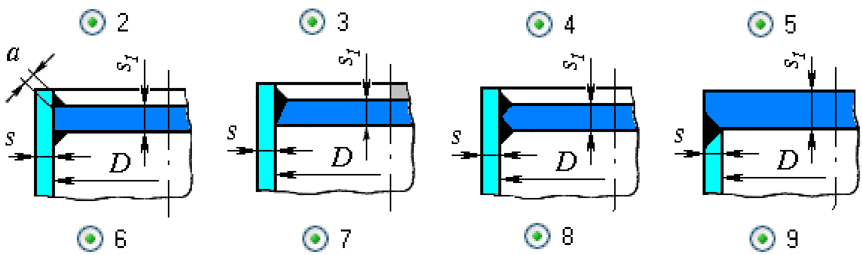


Fig. 3.45 Flat head

Component name, code of standards, material, dimensions, weld strength factors, insulation and lining, and load properties for flat heads are set in the same way as those for cylindrical shells. Head structure type is determined according to GOST 34233.2-2017 (see Fig. 3.46).



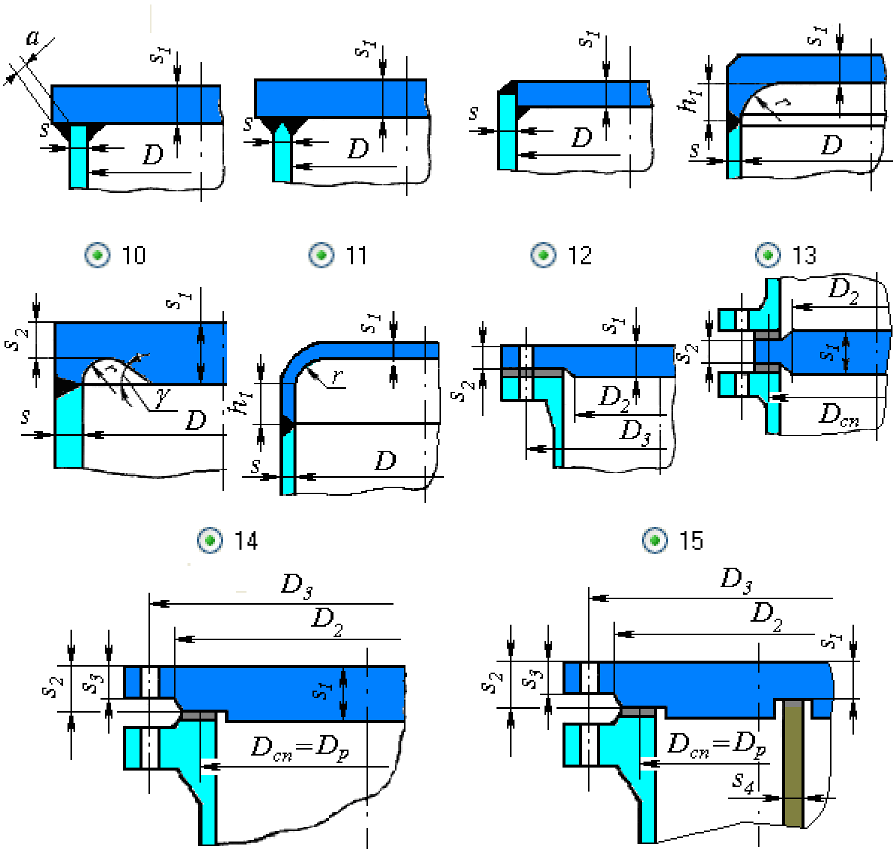
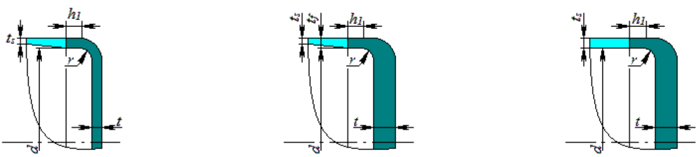
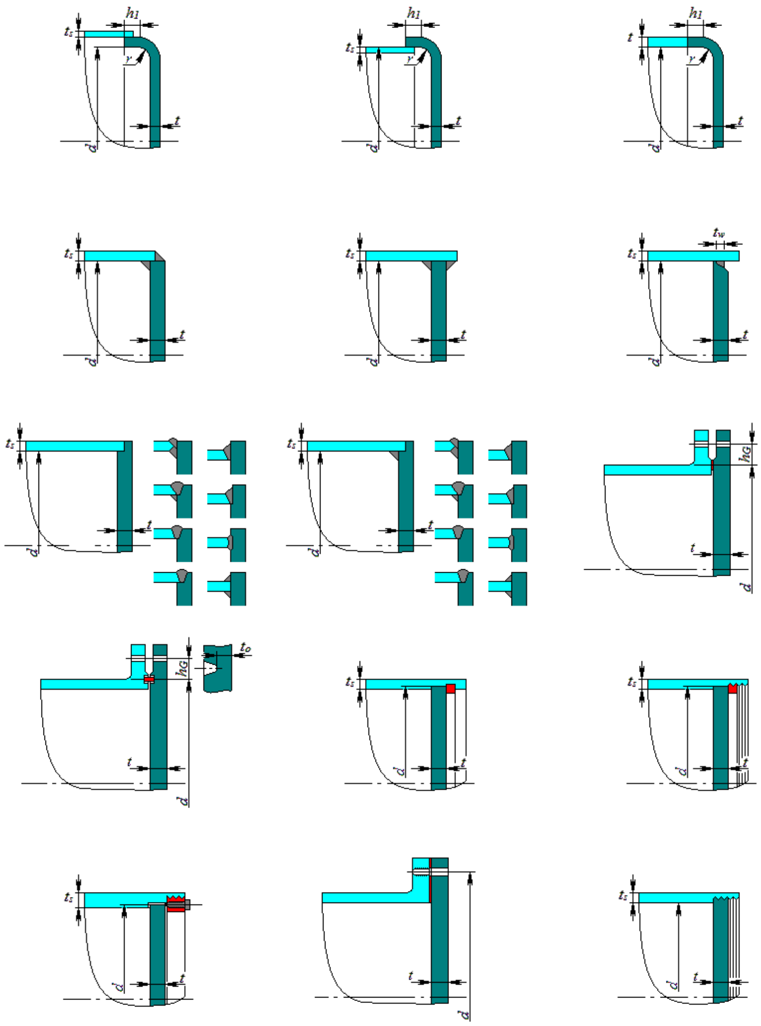


Fig. 3.46 Flat head types as per GOST

Head calculation is now available, as per ASME VIII-1 (versions of structure are indicated in Fig. 3.47).





h1
ts
t
r
d
tw
hG
to

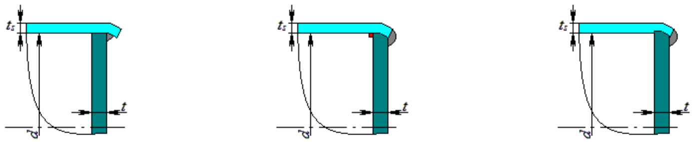


Fig. 3.47 Flat head types as per ASME

Note: don't use this component to model a tank supported by soil (the calculation method does not take into account the supporting conditions and in this case gives an excess margin).

3.17.8. Flat head with ribs

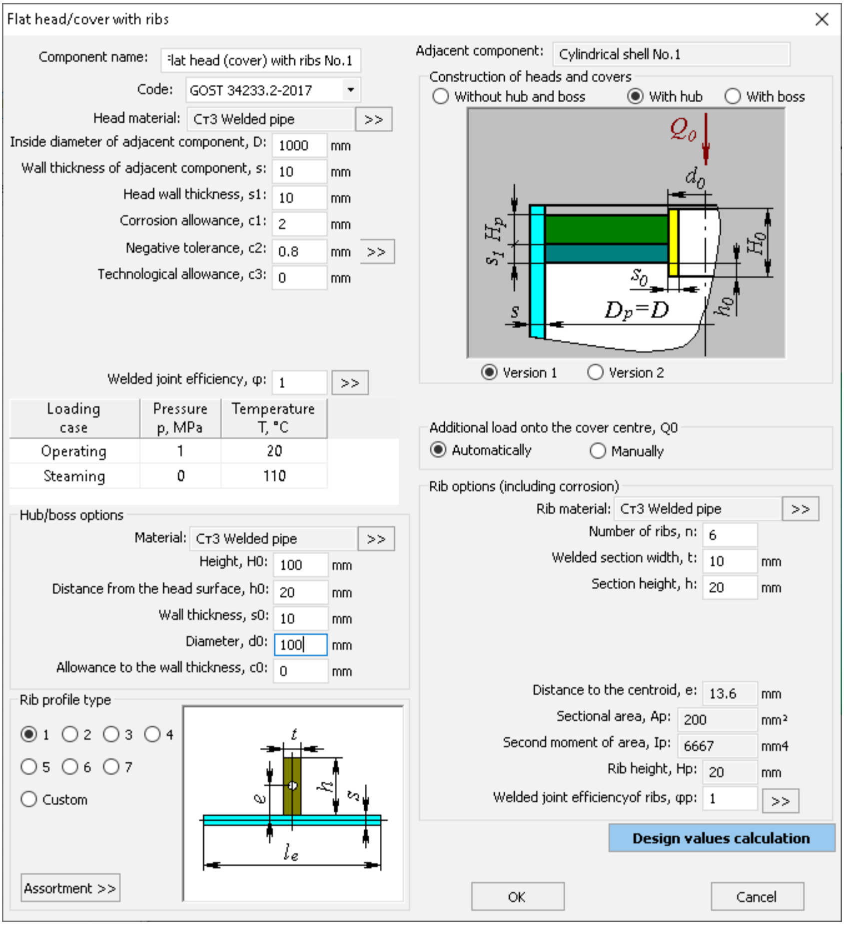


Fig. 3.48 Flat head with ribs

Rib section type and dimensions are set in the same way as those for stiffening ring of cylindrical shells.

It is possible to attach a child component (cylindrical shell) to the central part of the head, which will automatically determine the load on the center Q. If this load is applied manually, it is taken into account in the model and transferred to adjacent components.

The available options are shown on Fig. 3.49.

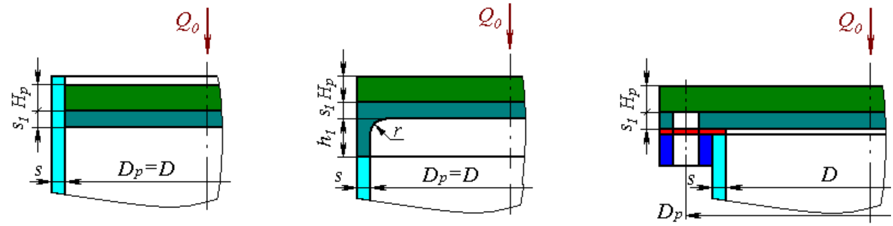


Fig. 3.49 Ribbed head types

3.17.9. Integral flat heads with opening

This component behaves in the construction like a conical transition (creates a difference in diameters). Adjacent elements can be attached to a smaller diameter.

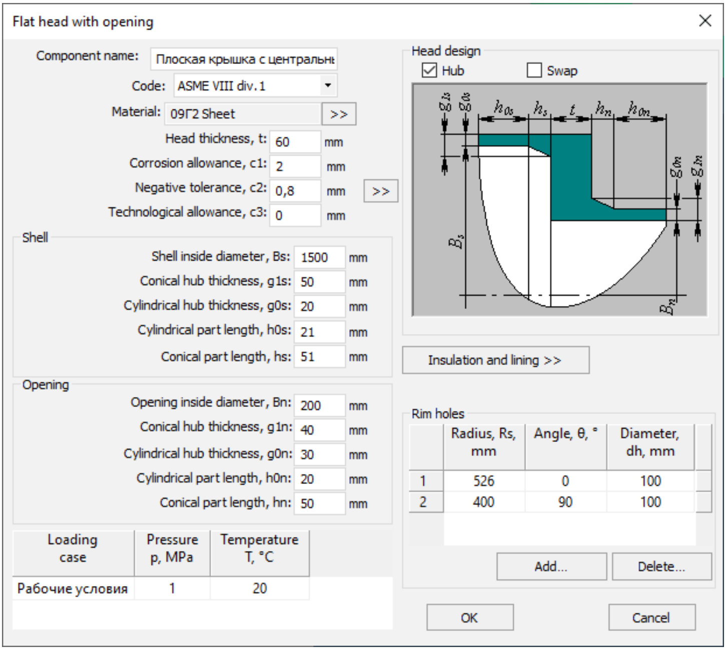
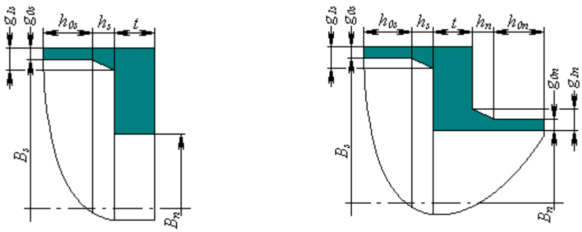


Fig. 3.50 Integral flat head with opening

The “Hub” option sets the hub on a smaller diameter (Fig. 3.51)

The “Swap” option sets the orientation of the component along the axis of the vessel.



a) without hub b) with hub

Fig. 3.51 Flat heads with opening designs

The “Rim holes” option allows to take into account the weakening by perforation (additional small holes located in the edge zone of the head).

3.17.10. Oval head

This component can be attached to oval nozzle.

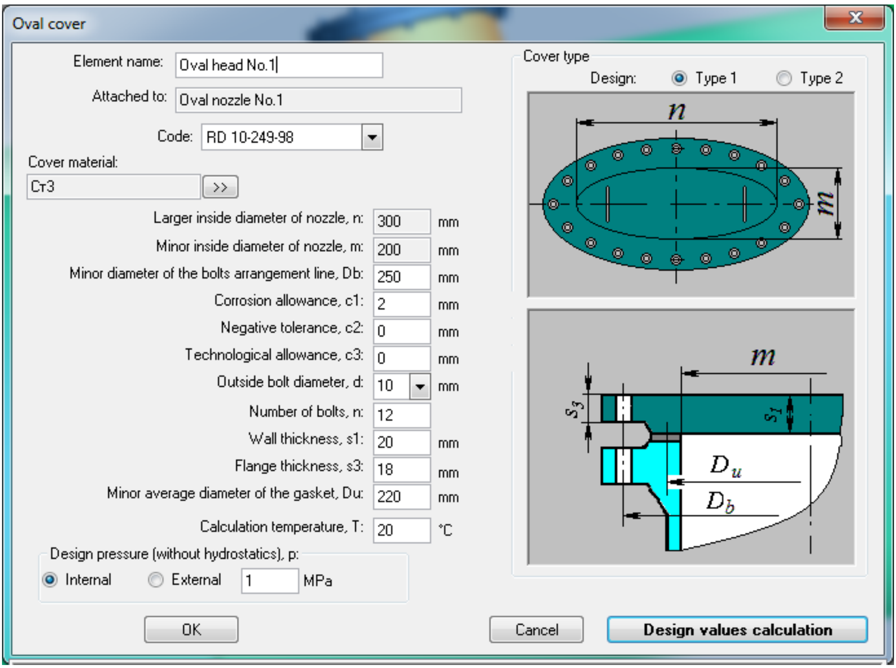


Fig. 3.52 Oval head

Possible designs of the head according to RD 10-249-98 shown in Fig. 3.53.

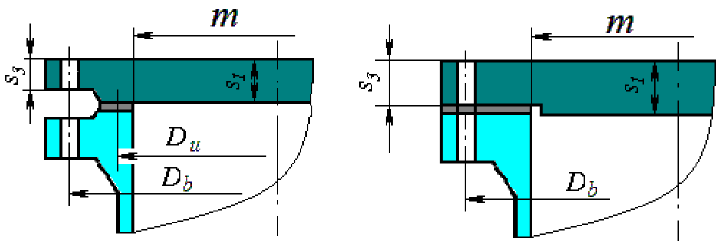


Fig. 3.53 Oval head types

3.17.11. Spherical head without knuckle

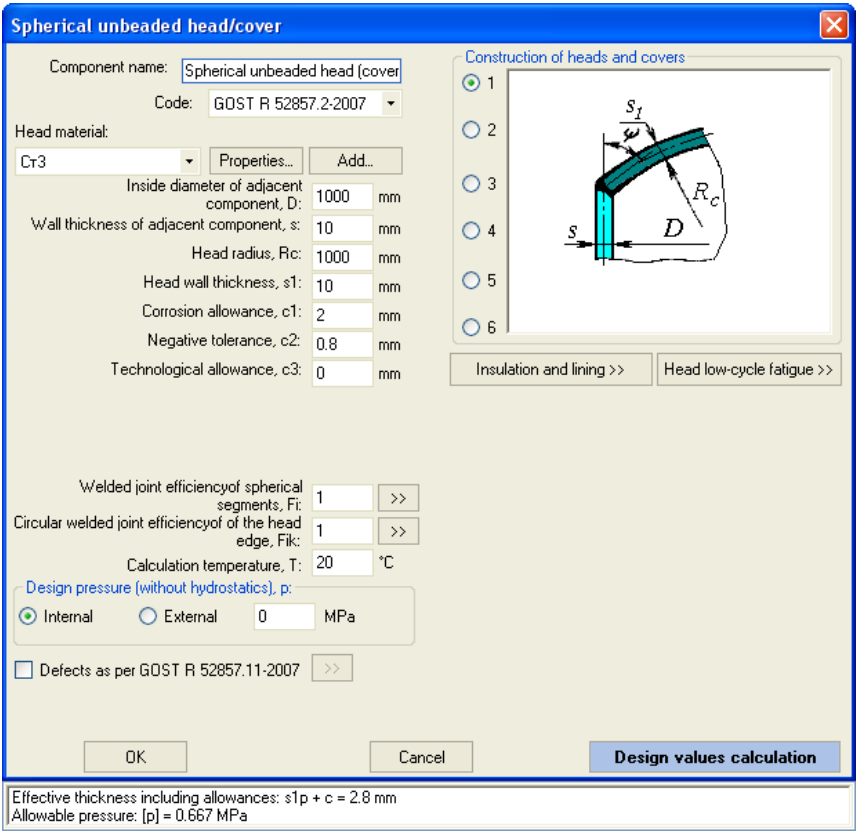


Fig. 3.54 Spherical head without knuckle

Component name, code of standards, material, dimensions, weld strength factors, insulation and lining, and load properties for flat heads are set in the same way as those for cylindrical shells. Head structure type is determined according to GOST 34233.2-2017 (see Fig. 3.55).

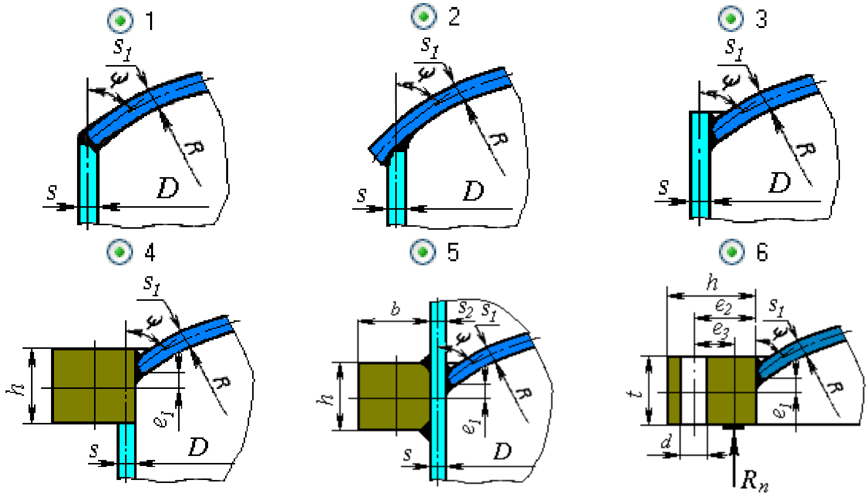


Fig. 3.55 Spherical head types

3.17.12. Nozzle

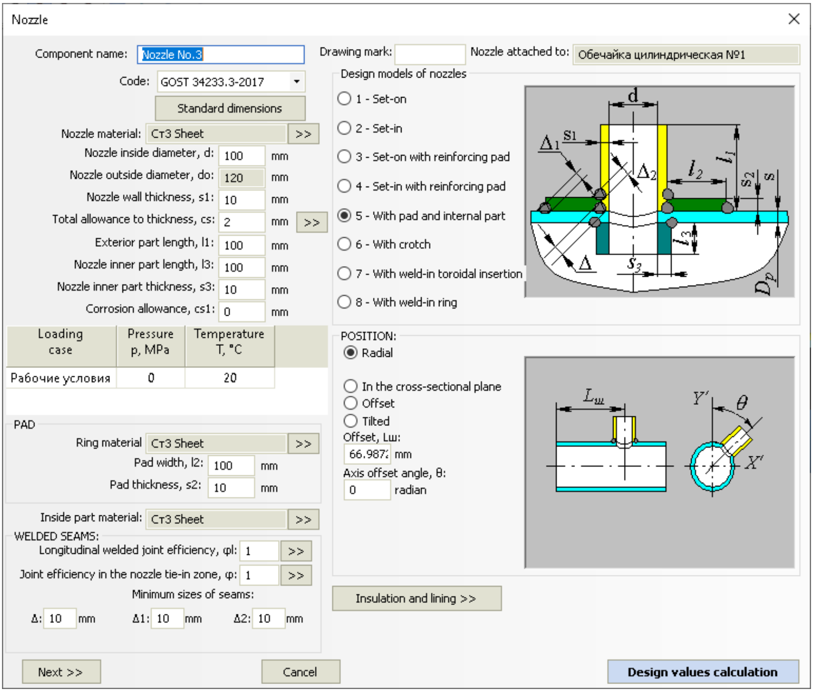
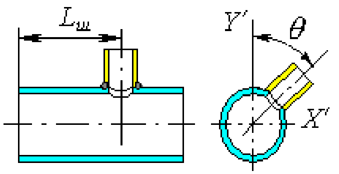
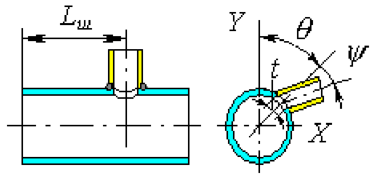


Fig. 3.56 Nozzle

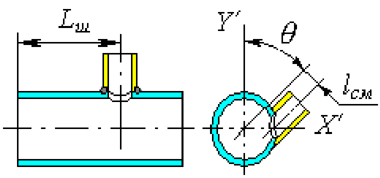
Component name, code of standards, material, dimensions, and weld strength factors for nozzles and padding ring (if present), as well as load properties, are set in the same way as those for cylindrical shells. Nozzle placement is determined based on the type of adjoining component.For cylindrical and conical shells and for conical heads, the nozzle can be radial (Fig. 3.57, a), positioned in the cross-sectional plane (Fig. 3.57, b), offset (Fig. 3.57, c), or placed arbitrarily (Fig. 3.57, d).



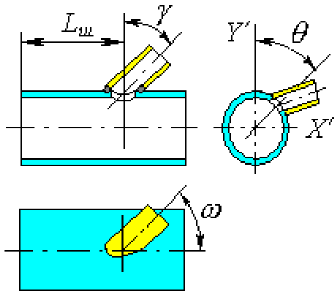
(a) Radial



(b) In the cross-section plane



(c) Offset



(c) Tilted

Fig. 3.57 Nozzle positioning on the cylinder

For dished heads (including spherical without knuckle), nozzle can be set in the polar or Cartesian coordinate system and can be radial, positioned along vessel's axis or positioned arbitrarily (Fig. 3.57). For flat heads, nozzles must be placed perpendicular to surface.

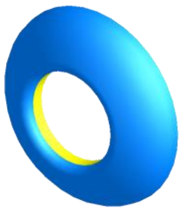


Fig. 3.58 Inward forming nozzle

For a set-in nozzle with the inner surface of the shell, select the "Set-in " configuration and set l_3=0.

For an inward forming nozzle (Fig. 3.58) set a negative value of “x”.

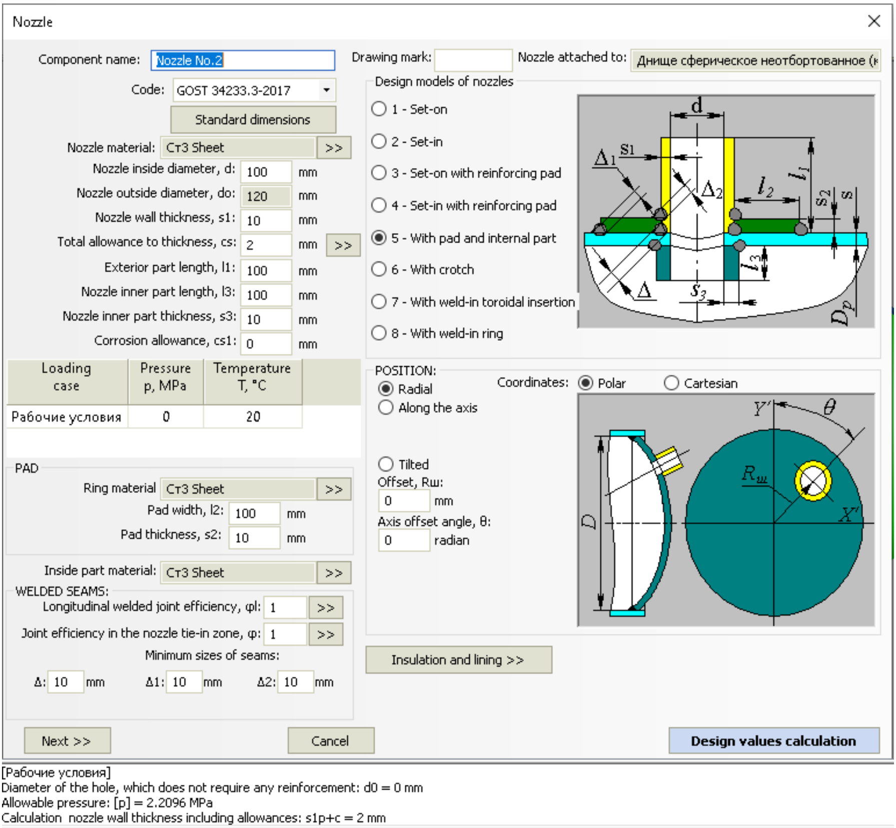
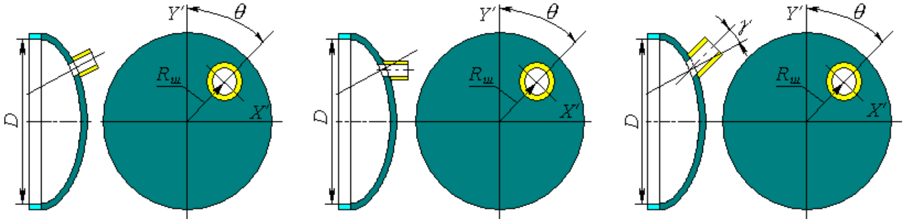


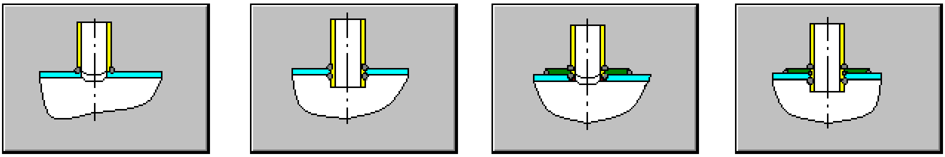
Fig. 3.59 Nozzle on the head



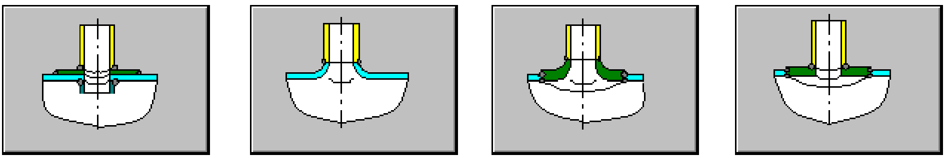
Radial   Along the axis   Tilted

Fig. 3.60 Nozzle positioning on the head

Nozzle model is determined according to GOST 34233.4-2017. See Fig. 3.61 for possible models.



Without additional reinforcement	With projecting inward	With pad	With projecting inward and pad



With pad and internal part	With beading	With toroidal insertion	With insert ring

Fig. 3.61 Nozzle types

An analysis of insertion point strength from external forces and moments (assigned using the Next >> button) is available for radially placed nozzles in cylindrical and conical shells and dished heads.

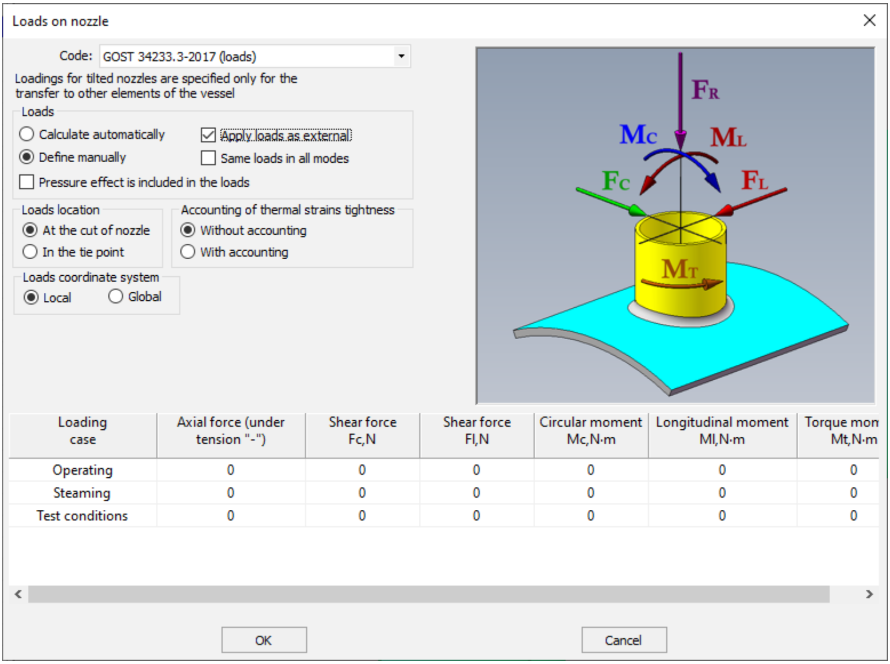


Fig. 3.62 Nozzle loads

In this case, in addition to nozzle reinforcement against pressure, an analysis of external forces and moments is carried out according to the selected standard: GOST 34233.3-2017, RD 26.260.09-92, WRC 537(107) /297, , EN 13445-3

Loads can be determined automatically during analysis based on the adjoining component or set manually. If “Apply as external” option is selected, input loads on the nozzle will be transferred to all model components.

The “Same loads in all modes” option allows you to avoid filling the whole table of loads individually for each mode if the loads are the same or differ slightly.

Using the “Loads coordinate system” option, loads can be specified in the nozzle coordinate system (“Local”) or in the model coordinate system (“Global”).

When manually assigning loads, a user can also specify, at what point they are applied (option "Location of loads"). When assigning loads on the nozzle cut, they are automatically recalculated during the calculation, taking into account length l_1.

For flat head, operability under pressure is evaluated considering presence of passages.

Pay attention to positive and negative signs when setting forces and moments. Positive values correspond to directions indicated on the model. Analytical model displayed in Fig. 3.62 is applicable only for radial nozzles. For other variants of structure it is necessary to control the direction of loads on the displayed model, as in general the coordinate system of nozzle is set at an angle θ first, then at ω, and then at γ or ψ.For instance, a shifted nozzle is resulted from the tilted nozzle, at ω = 90°.

At calculation of loads using FEM method, cut-in is represented as several beam elements (Fig. 3.63):

- Element marked with blue joins outer wall of supporting shell with axial line of the shell in the cut-in point. This element is a rigid link.
- Chain of elements marked with red is modelled by weightless ring cross-section beam elements. Weight load is applied to the yellow node placed in the gravity centre of nozzle.

External loads are applied to point 1 or 2, depending on the selected position.

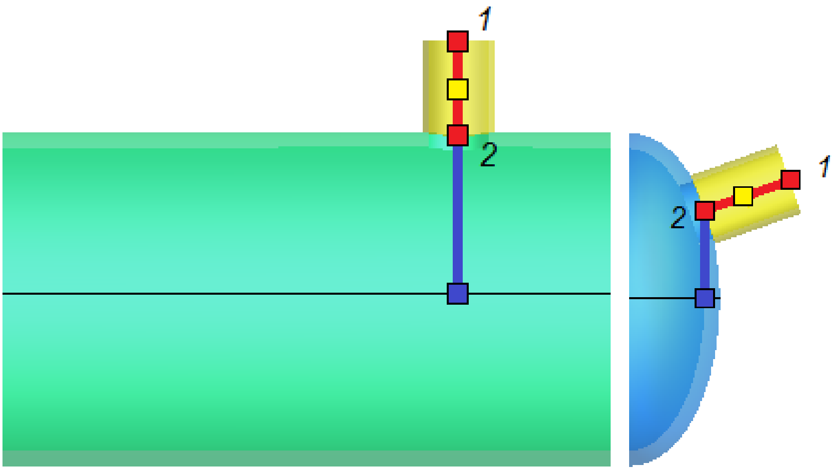


Fig. 3.63 Modeling a nozzle with beam elements

3.17.13. Oval nozzle

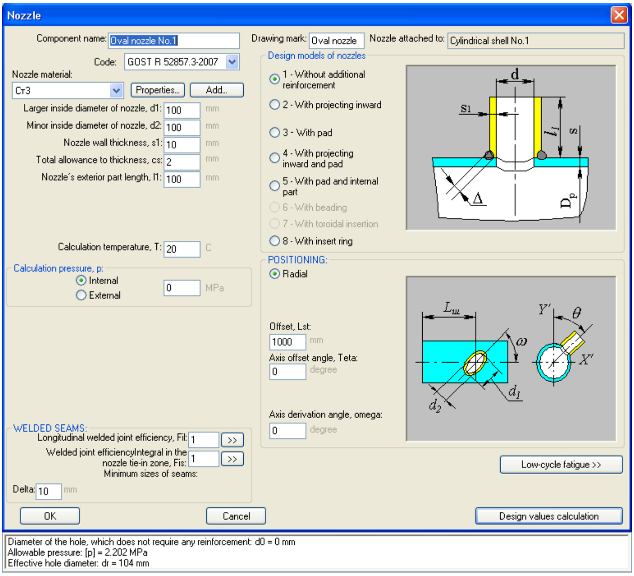


Fig. 3.64 Oval nozzle

Component name, material, dimensions, weld strength factors for nozzle and padding ring (if present), load properties and location are set in the same way as those for ordinary nozzles.

3.17.14. Bend

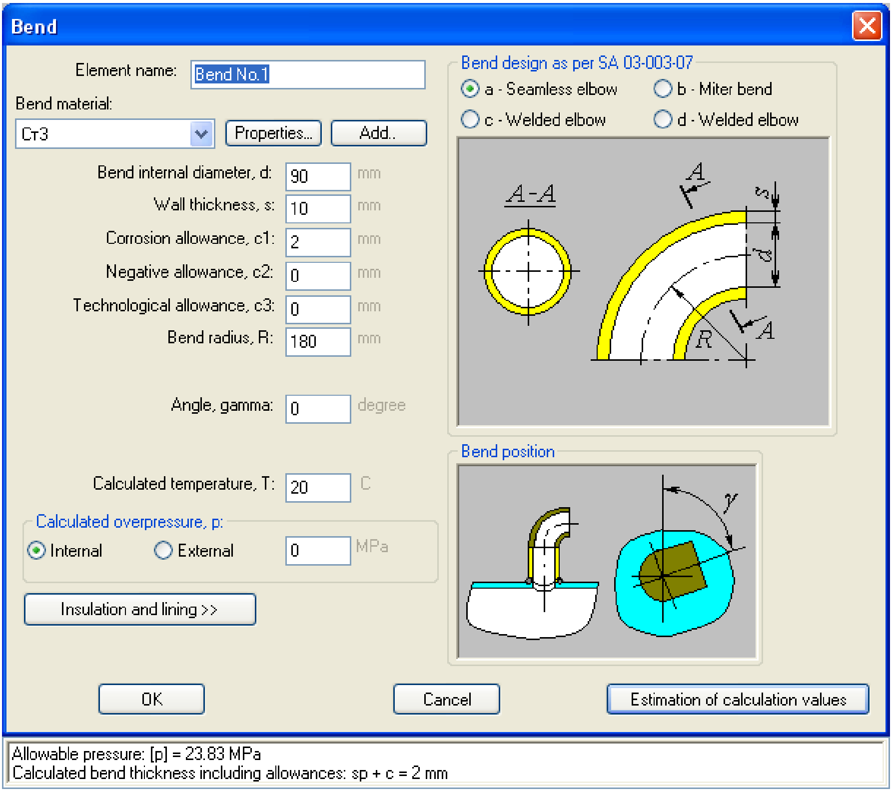
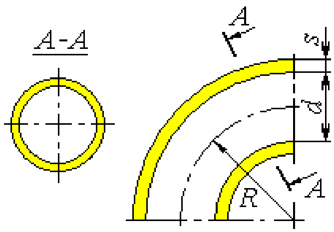


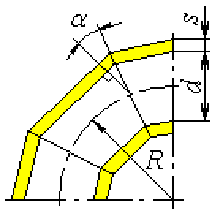
Fig. 3.65 Bend

Component name, material, dimensions, weld strength factors and load properties are set in the same way as those for cylindrical shells. Bends are connected to nozzles and their adjoining shells. Bend placement is determined by its bend angle.

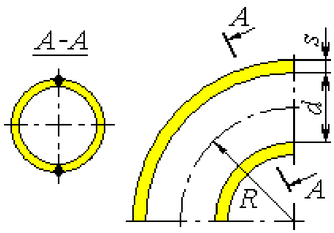
Bend structure is determined according to SA 03-003-07. Bends can be seamless (Fig. 3.66,a), sectorial (Fig. 3.66,b), welded longitudinally (Fig. 3.66,c) and welded transversely (Fig. 3.66,d).



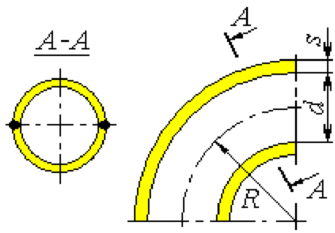
(a) Seamless



(b) Sectorial



(c) Welded, when the welds are positioned in the curve plane



(d) Welded, when the welds are positioned along the neutral line

Fig. 3.66 Bend types

3.17.15. Flange joint

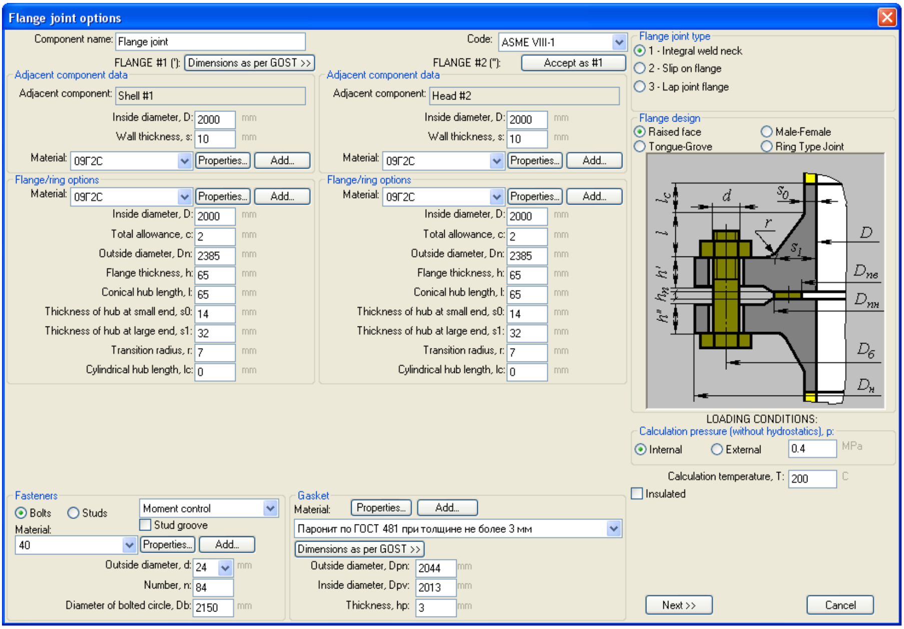


Fig. 3.67 Flange joint

Calculation of flange joints is possible as per RD 26-15-88, GOST 34233.4-2017, ASME VIII div.1, ASME VIII div.2. Comparison of codes for consideration of loads is specified below:

Table 3-4

Code	Pressure consideration	Consideration of external loads (F, M)	Consideration of temperature loads
RD 26-15-88	√	√	√
GOST 34233.4-2017	√	√	√
ASME VIII div.1	√	–	–
ASME VIII div.2	√	√	–

Component name, code of standards, material, dimensions and load properties for flanges are set in the same way as those for cylindrical shells. Flange type is determined according to GOST 12820(12821,12822)-80 (see Fig. 3.68).

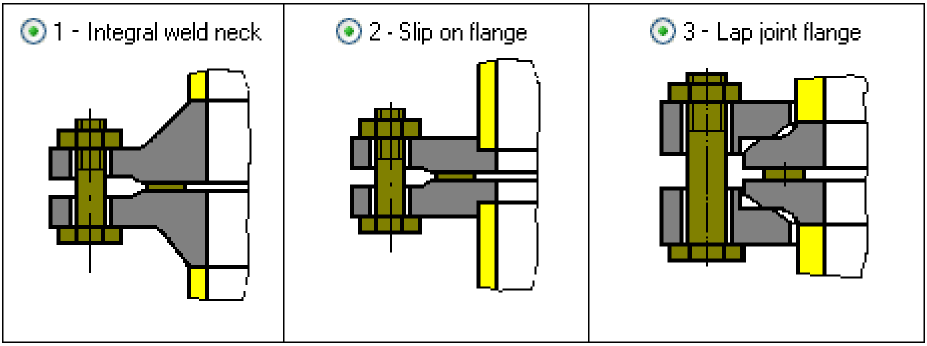
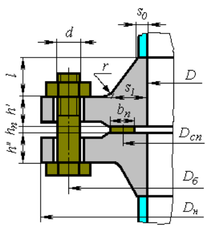
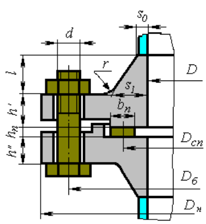


Fig. 3.68 Flange joint types

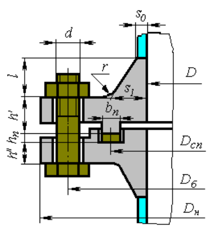
Fig. 3.69 - Fig. 3.73 show various flange joint models as per GOST 34233.4-2017.



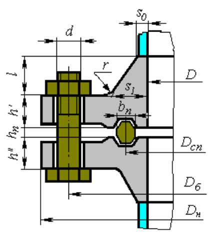
(a) Raised face



(b) Male-Female

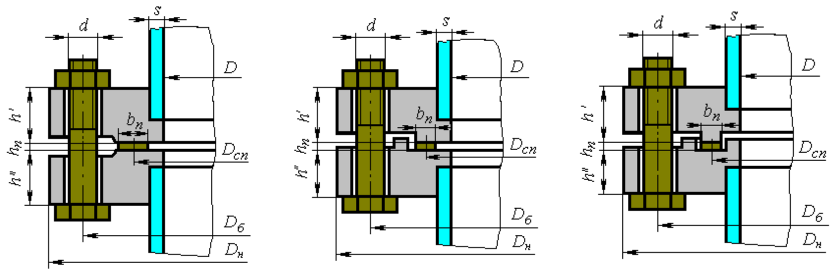


(c) Tongue-Grove



(d) Ring Type Joint

Fig. 3.69 Butt-welded flanges according to GOST 28759.3-90 (a,b,c) and GOST 28759.4-90 (d)



(b) Raised face (b) Male-Female (c) Tongue-Grove

Fig. 3.70 Flat welded flanges according to GOST 28759.2-90

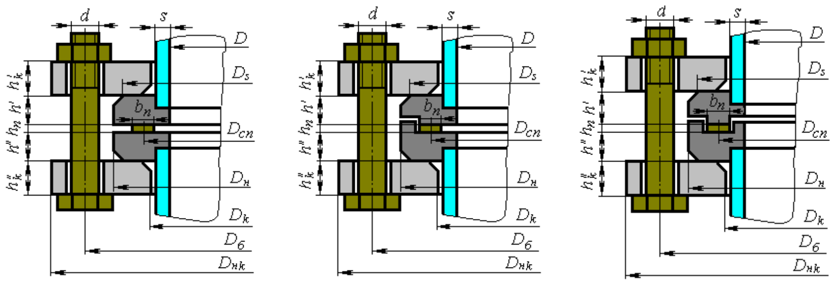
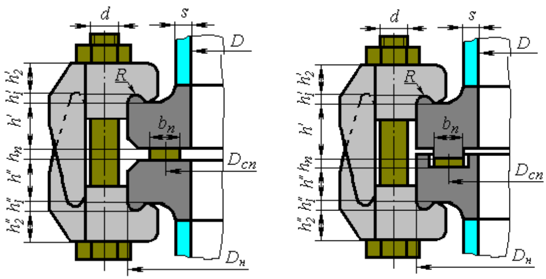
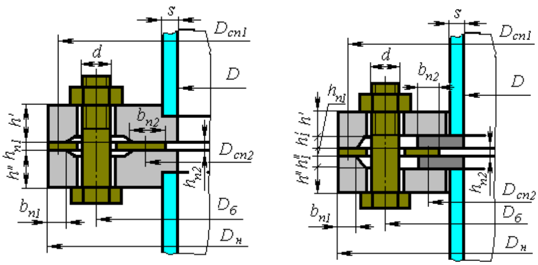


Fig. 3.71 Flanges with free rings



(a) Flat face (b) Tongue-Grove

Fig. 3.72. Flanges for fasteners according to OST 26-01-396-78



(a) (b)

Fig. 3.73 Contact flanges

Surface design of the flange joint at calculation as per ASME VIII-1(2) is shown in Fig. 3.74.

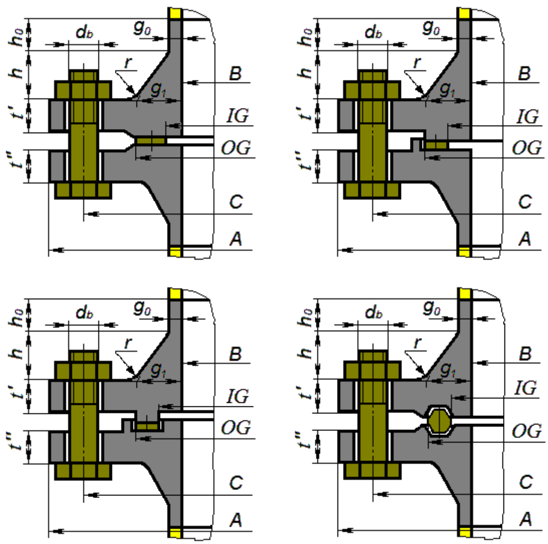


Fig. 3.74 Flange joint types as per ASME

Size of flanges, fasteners and gaskets can be selected from the database of standard parts by selecting the flange joint type and variation and pressing the "Standard dimensions" button. These properties will be set for both flanges.

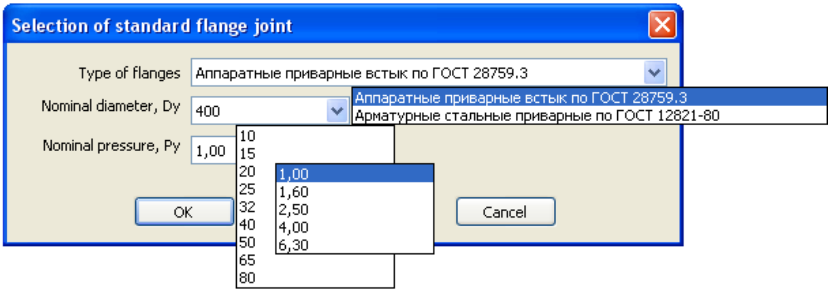


Fig. 3.75 Standard flange joint selection

To select a gasket from the database, its material must first be selected. Gasket type must match flange type to allow the selection of a standard part.

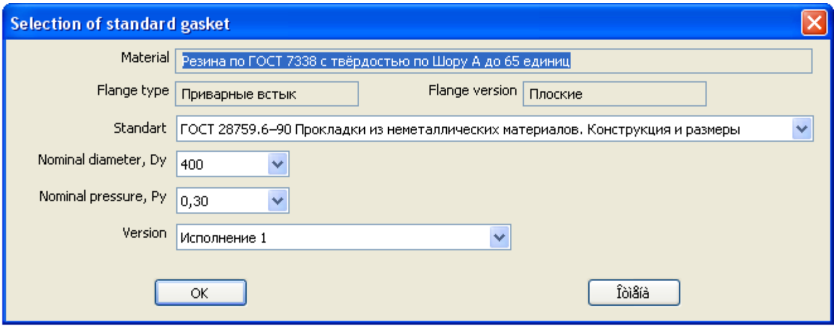


Fig. 3.76 Standard gasket selection

Dialog window (Fig. 3.76) may differ from the example, as it is determined by the flange joint type, its variation and the gasket material selected.

Bolt (stud) materials and properties can be selected from the database or input manually.

Pressing key 

, you can open a dialog of extended properties of flanges and fasteners (Fig. 3.77):

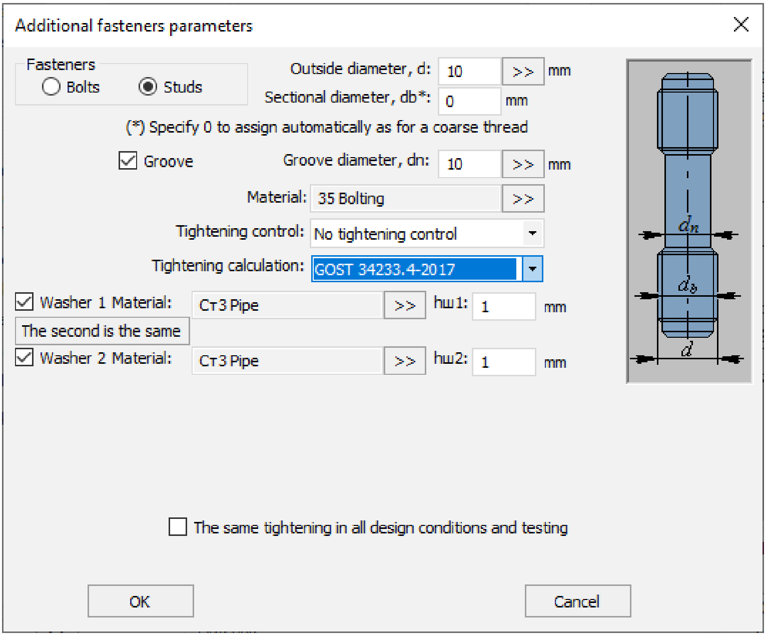


Fig. 3.77 Fasteners additional parameters

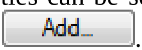
Item **Groove** is used for rods with groove diameter less than the internal threading diameter. Flange insulation (

) affects the temperature of flange joint components.

Selection of option «**Control per moment**» activates checkbox «**Calculation of bolting load without allowance for minimum initial tension of bolts (0.4*[б]*Ab)**», in some cases it helps avoiding of excessive reinforcement of flanges.

If you select item "Uniform tightening under operating conditions and tests", bolting load will be taken the same (maximum of all) for all modes.

The “Tightening calculation” option allows you to select an alternative standard for calculating the torque on the wrench.

Flange joint gasket and its properties can be selected from the RD 26-15-88 database or input manually by pressing 

.

Database of materials of flanges and bolts is sensitive to the selected calculation code. This is due to the fact that ASME II Part D includes a large volume of data per allowable stresses, which can be used **only** in calculations as per ASME VIII-1(2).

Flange insulation influences on calculation temperatures of flange joint components, weight and material consumption.

Checkbox «Insertion» is used for the part clamped between the flanges (line-blanks, blind, etc.).

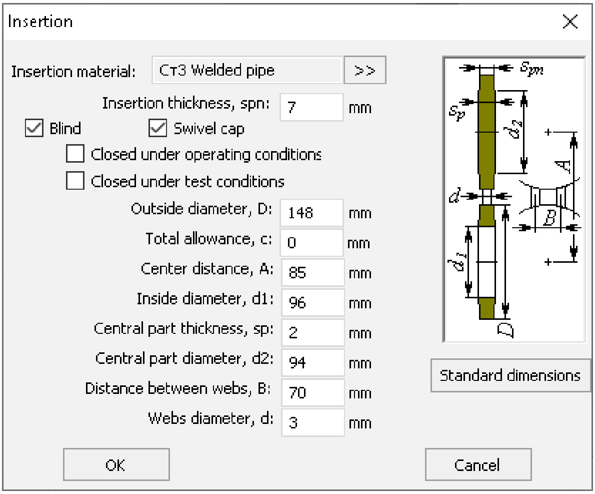


Fig. 3.78 Insertion

Item "More" allows more detailed sealing surfaces parameters (Fig. 3.79).

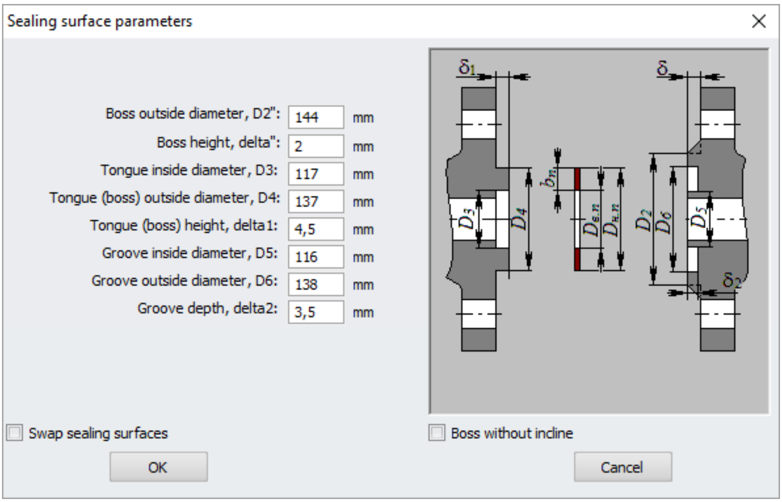


Fig. 3.79 Sealing surfaces

Option "Swap sealing surfaces" allows exchanging "male-female”.

The “Flange Thickness” parameter has a peculiarity — having an unsymmetrical configuration of the sealing surface (tongue-groove,, male-female), it can refer both to flange No. 1 and flange No. 2 (depending on the state of the option “Swap sealing surfaces”).

External forces and moments can be set by pressing 

.

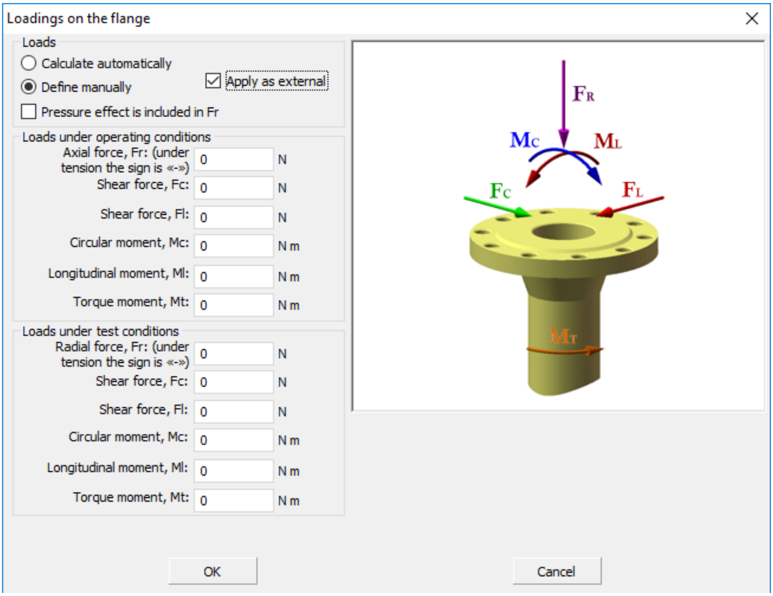


Fig. 3.80 Flange loads

Loads can be calculated automatically during analysis based on components adjoining the flange joint or set manually. If “Apply as external” option is selected, set loads on the flange joint will be transferred to all model components.

3.17.16. Reversal flange

A reversal flanges set similarly to the flange connection.

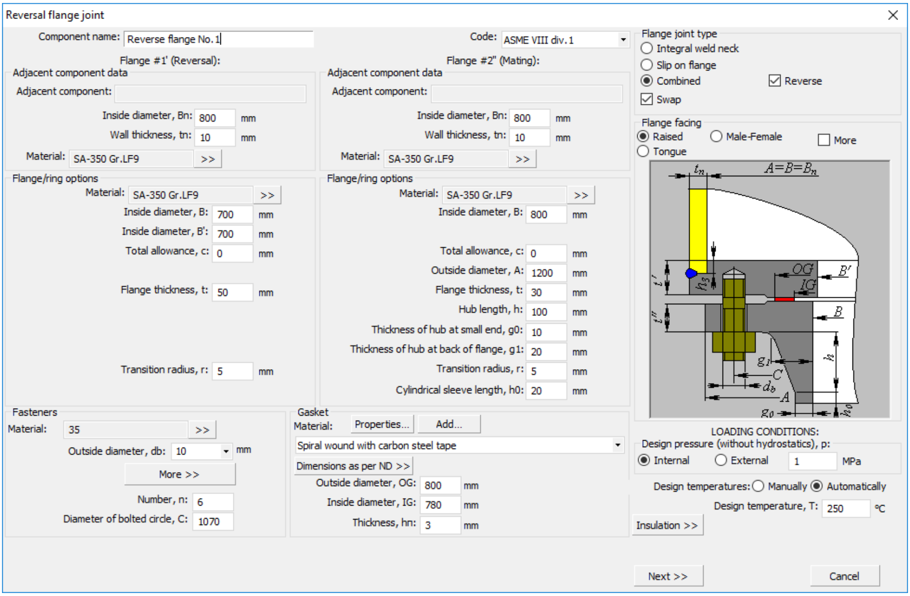
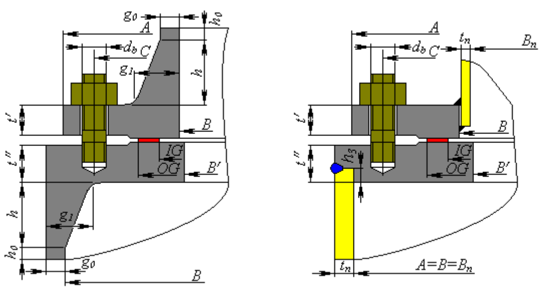


Fig. 3.81 Reversal flange

Calculation of reversal flanges can be done according to ASME VIII div.1. Available calculation schemes are shown in Fig. 3.84.



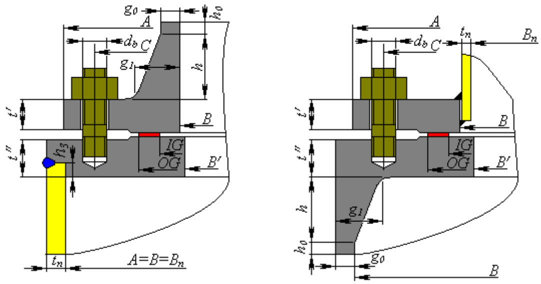


Fig. 3.82 Reversal flange types

3.17.17. Bolted heads

Bolted heads include three types – flat, ellipsoidal and spherical without knuckle. They are composed of a flange and the head itself.

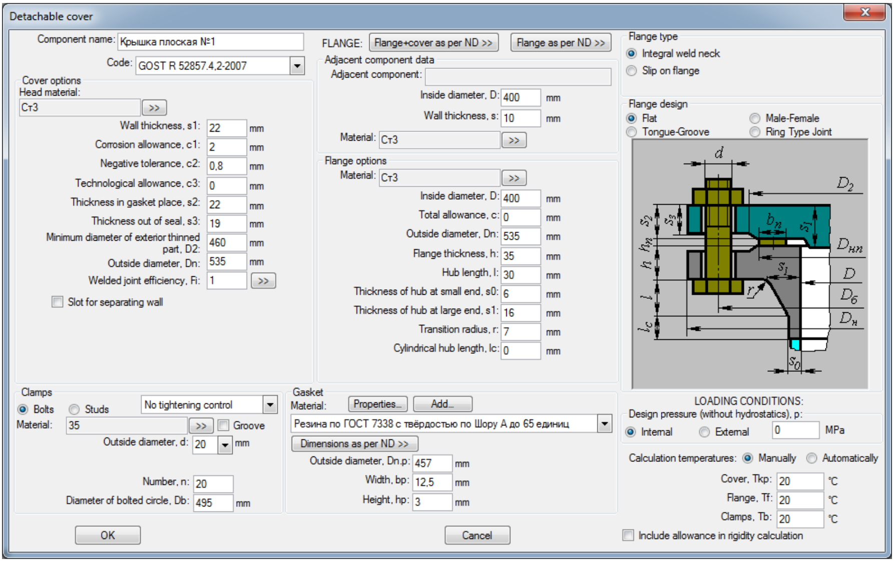
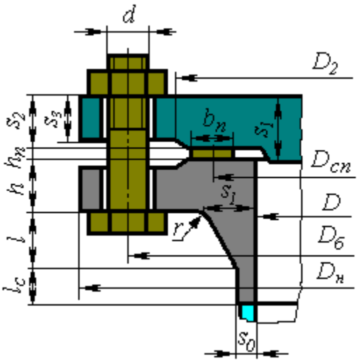
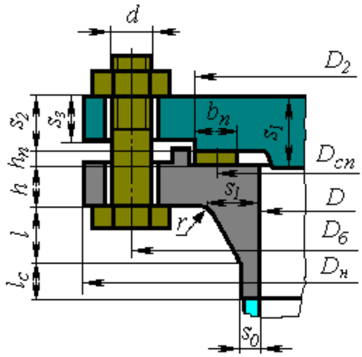


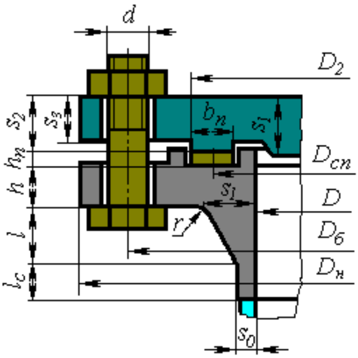
Fig. 3.83 Flat bolted head

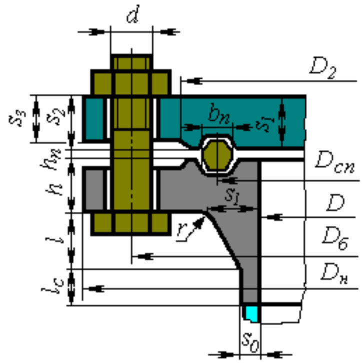
Bolted heads can be adjoined to the same components as welded heads.

There is a possibility of selecting either head assembled together with the flange (components are selected from database, so that the fasteners parameters match), or the flange separately (in case of non-standard heads).


d
D2
s2
s3
bп
s1
hп
Dсп
h
s1
D
r
Dб
l
Dн
lc
s0


d
D2
s2
s3
bп
s1
hп
Dсп
h
s1
D
r
Dб
l
Dн
lc
s0


d
D2
s2
s3
bп
s1
hп
Dсп
h
s1
D
r
Dб
l
Dн
lc
s0


d
D2
s3
s2
bп
s1
hп
Dсп
h
s1
D
r
Dб
l
Dн
lc
s0

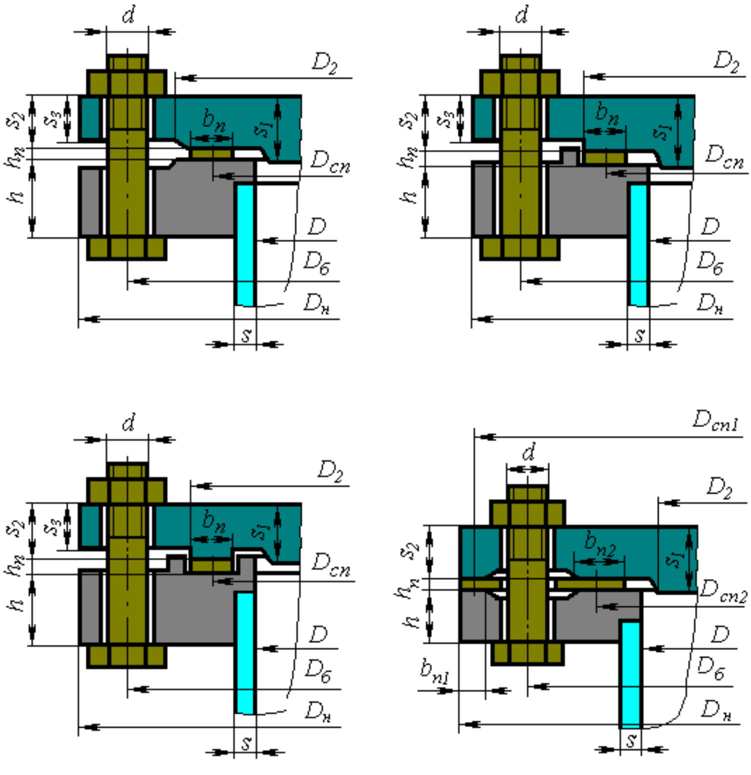
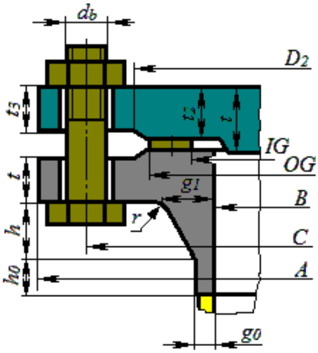
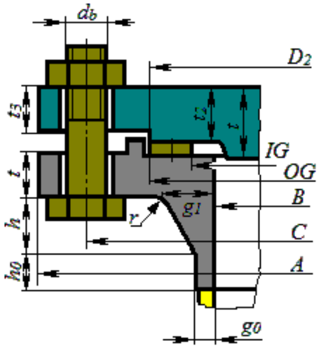
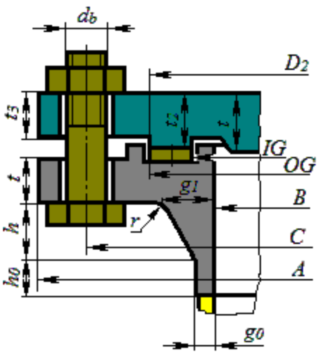
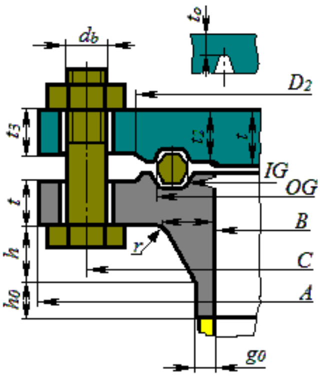


Fig. 3.84 Flat bolted heads as per RD 26-15-88, GOST 34233.4-2017


db
D2
t3
t2
t
IG
OG
g1
B
r
h
C
h0
A
g0


db
D2
t3
t2
t
IG
OG
g1
B
r
h
C
h0
A
g0


db
D2
t3
t2
t
IG
OG
g1
B
r
h
C
h0
A
g0


to
db
D2
t3
t2
t
IG
OG
B
r
h
C
h0
A
g0

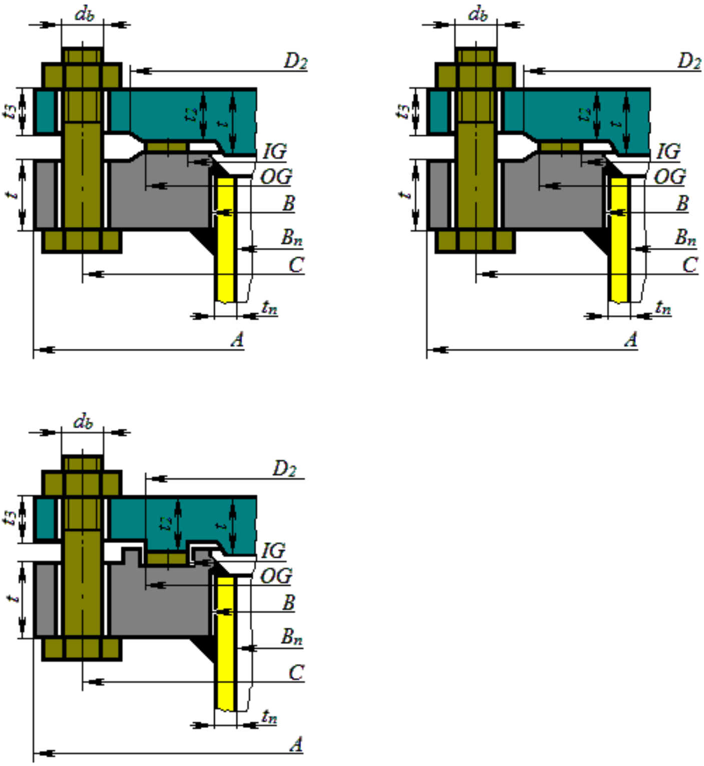


Fig. 3.85 Flat bolted heads as per ASME VIII-1

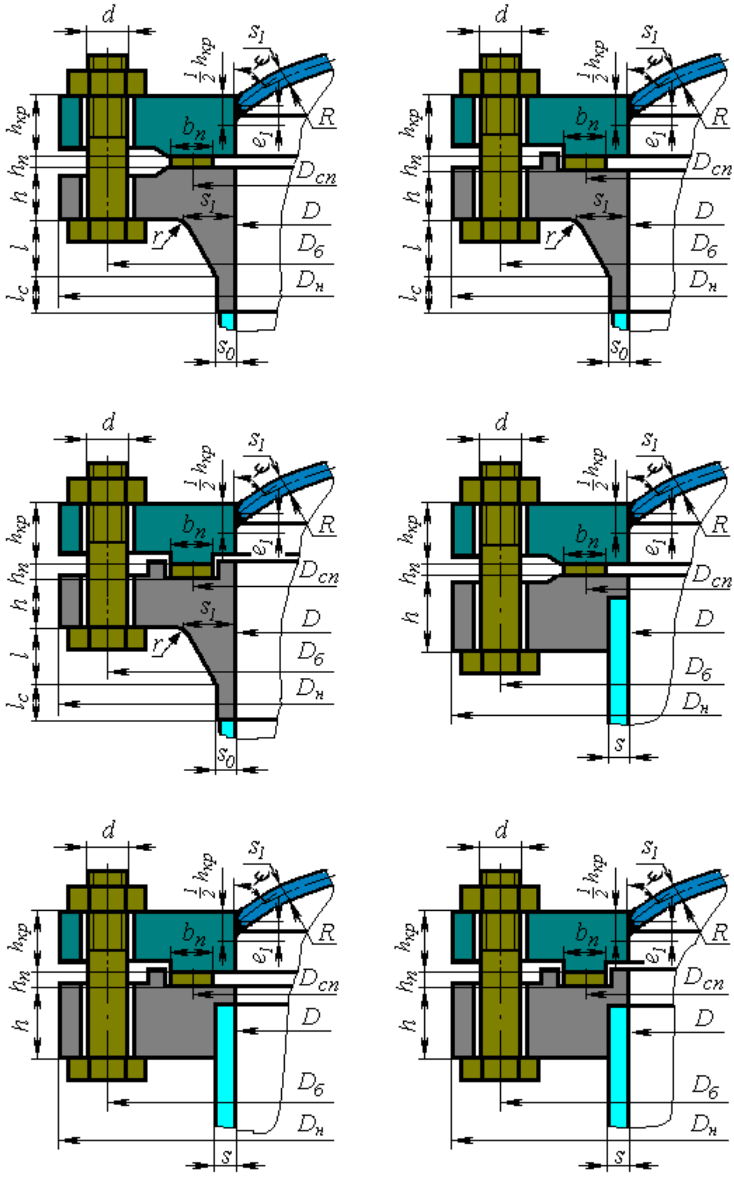
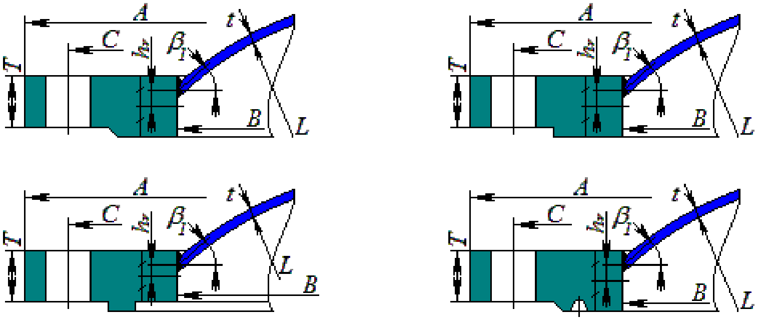
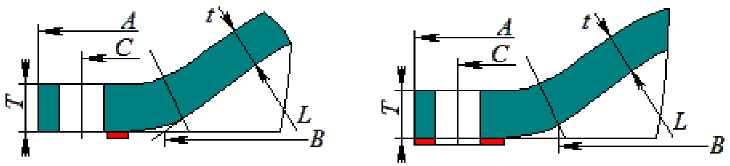


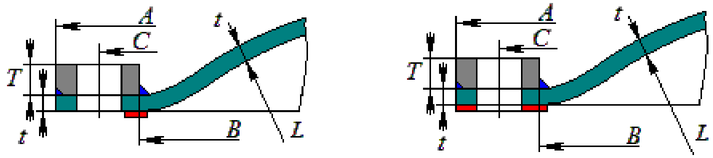
Fig. 3.86 Spherical bolted heads as per RD 26-15-88, GOST 34233.4-2017



1-6 (b)

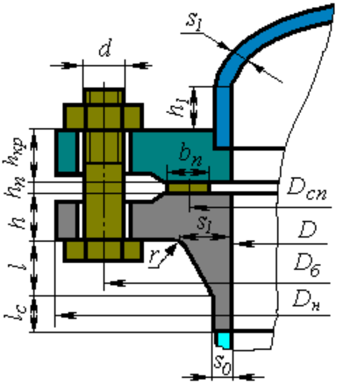


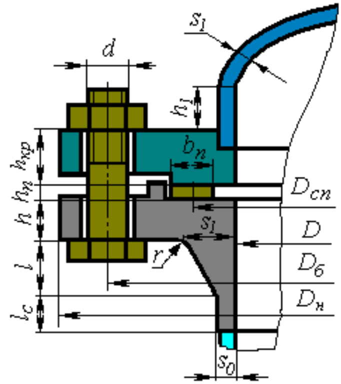
1-6 (c)

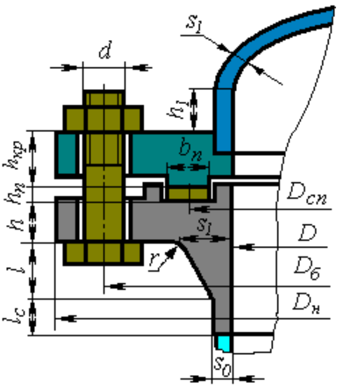


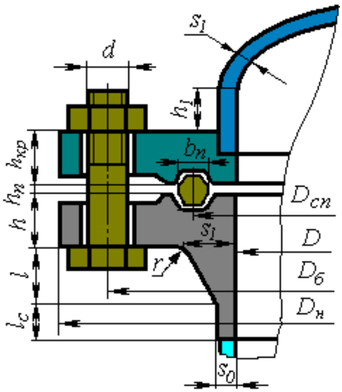
1-6 (d)

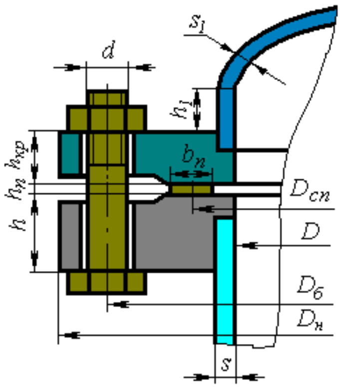
Fig. 3.87 Spherical bolted heads as per ASME VIII-1

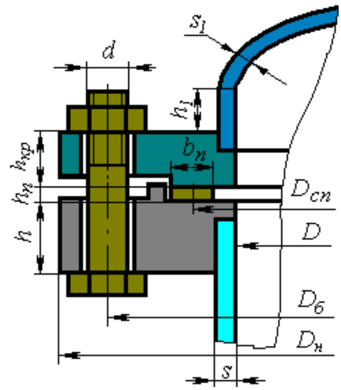

s_1
d
h_1
$h_{кр}$
$b_п$
$h_п$
$D_{сп}$
h
s_1
D
r
l
$D_б$
l_c
$D_н$
s_0


s_1
d
h_1
$h_{кр}$
$b_п$
$h_п$
$D_{сп}$
h
s_1
D
r
l
$D_б$
l_c
$D_н$
s_0


s_1
d
h_1
$h_{кр}$
$b_п$
$h_п$
$D_{сп}$
h
s_1
D
r
l
$D_б$
l_c
$D_н$
s_0


s_1
d
h_1
$h_{кр}$
$b_п$
$h_п$
$D_{сп}$
h
s_1
D
r
l
$D_б$
l_c
$D_н$
s_0


s_1
d
h_1
$h_{кр}$
$b_п$
$h_п$
$D_{сп}$
h
D
$D_б$
$D_н$
s


s_1
d
h_1
$h_{кр}$
$b_п$
$h_п$
$D_{сп}$
h
D
$D_б$
$D_н$
s

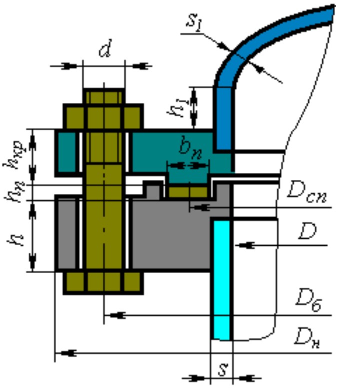
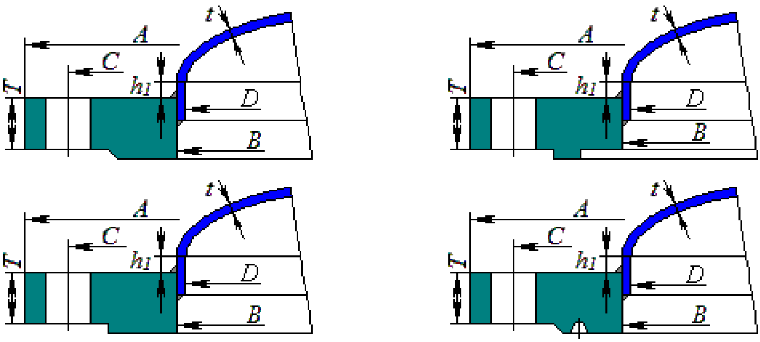
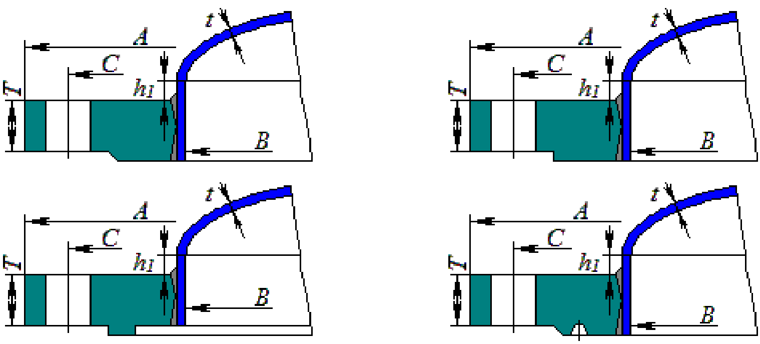


Fig. 3.88 Elliptic bolted heads as per RD 26-15-88, GOST 34233.4-2017



1-6 (a) with partial penetrationc



1-6 (a) with full penetration

Fig. 3.89 Elliptic bolted heads as per ASME VIII-1

3.17.18. Stiffening ring

Stiffening rings can be adjoined to any cylindrical shells in the model. Component name, material, dimensions, weld strength factors and load properties for stiffening rings are set in the same way as those for cylindrical shells. Ring placement on the model is determined by the adjoining component and the distance from the left (head) margin (toward Z-axis). The ring can be placed both inside and outside the shell.

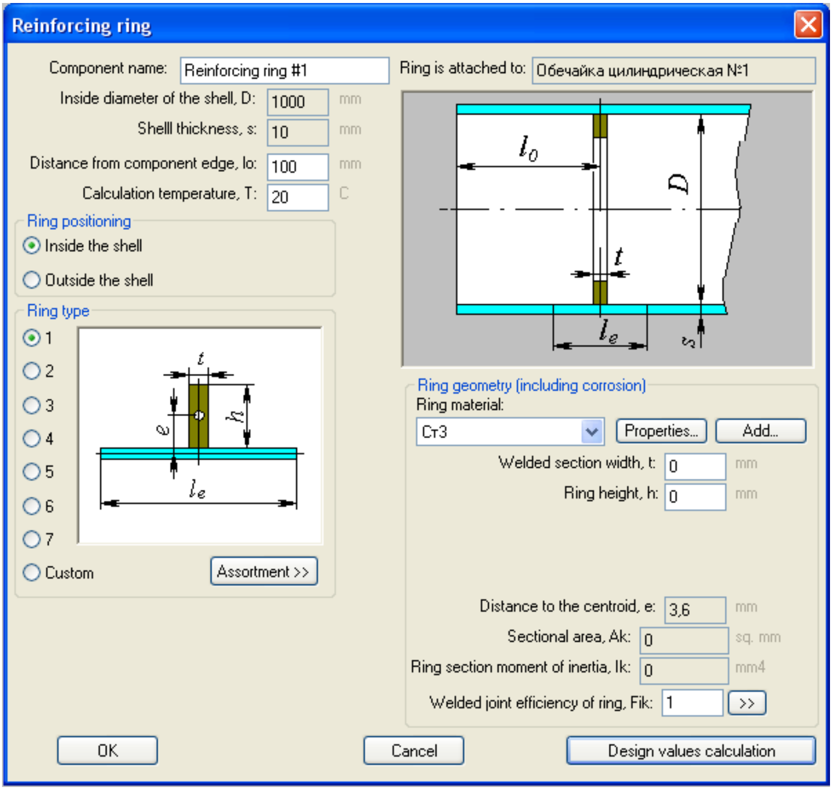


Fig. 3.90 Stiffening ring

Ring type and dimensions are determined by standard cross-sections or input manually. Material corrosion must be taken into account.

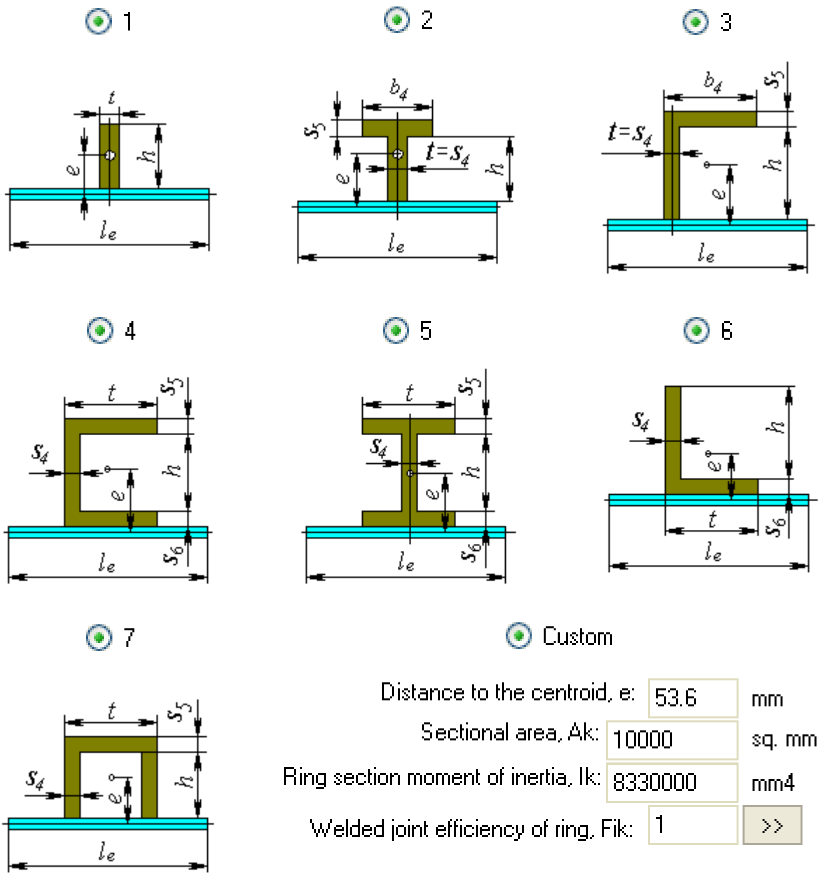


Fig. 3.91 Standard sections

Standard cross-section of selected pipe can be selected from the database using the [Assortment >>] button.

3.17.19. Stiffening rings group

This component provides setting of group of stiffening rings of the same section, located at regular intervals. In calculating, each ring within the group is considered individually. So, the groups of rings can be combined with single rings. These rings are set similarly to the component stiffening ring.

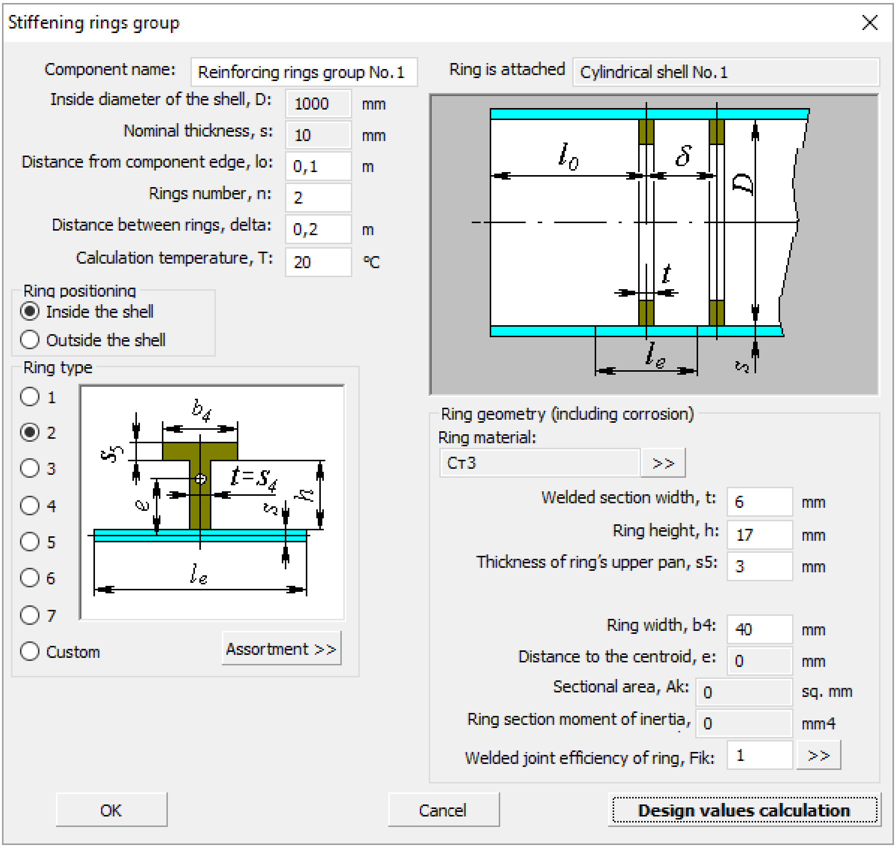


Fig. 3.92 Stiffening rings group

3.17.20. Saddle support

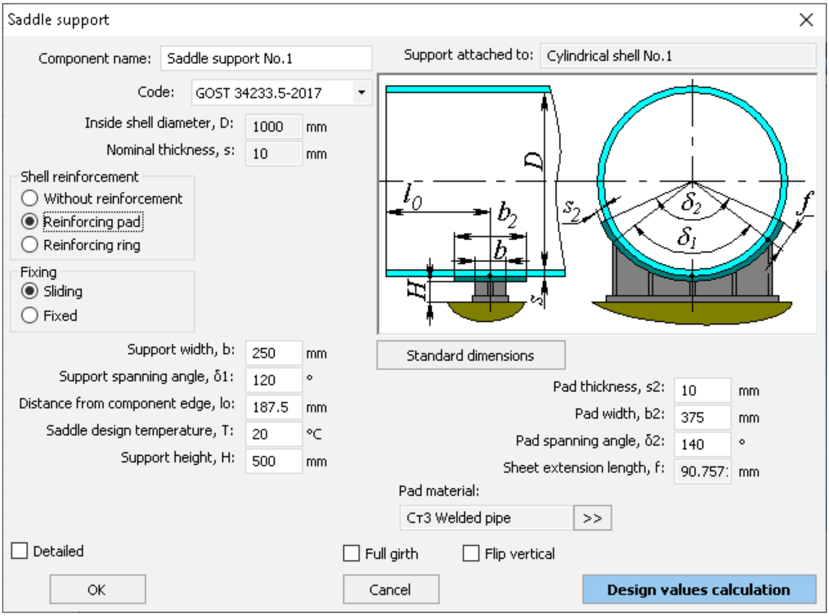


Fig. 3.93 Saddle support

Saddle supports can be adjoined to any cylindrical shells of the vessel casing. Its placement and dimensions determine the analysis of bearing loads on vessel components. The number of supports must be no less than two.

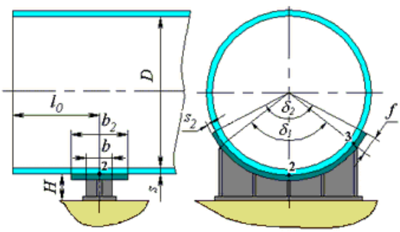


Fig. 3.94 Saddle support with reinforcing pad

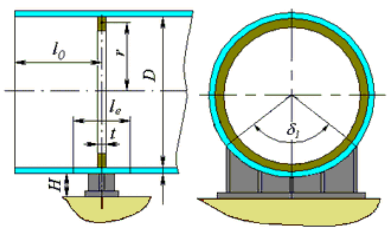


Fig. 3.95 Saddle support with reinforcing ring

Name, code of standards and dimensions of saddle supports are set in the same way as those for cylindrical shells.

A saddle support can have no reinforcements or be supported by a reinforcing pad or stiffening ring.

When supported by stiffening ring, the ring's type, placement and dimensions are set in the same way as those for cylindrical shells (see item 3.17.18). The ring is considered at calculation of cylindrical shell against pressure influence. A user can also consider presence of spacing ribs in the internal stiffening ring (Fig. 3.96). Cross-section parameters of spacing rib can be set by selecting  key.

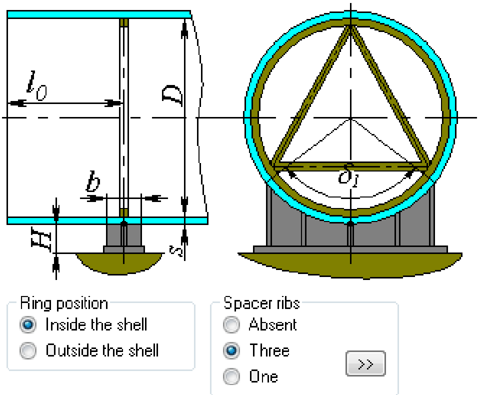


Fig. 3.96 Spacer ribs inside of ring

The "Full girth" option affects the visual display of the support and allows to form a full girth support consisting of two components (in the second component, the option "Flip vertically" must be enabled).

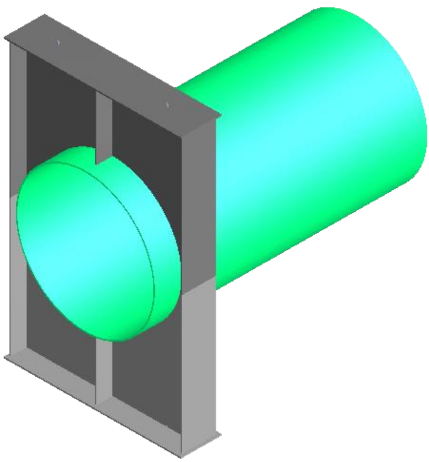


Fig. 3.97 Full girth support consisting of two components

One of the vessel supports must be fixed.

At calculation of loads using FEM method, a saddle support is represented as two beam elements (Fig. 3.98):

- Element marked with red connects the pinning point with outside wall of supporting shell. This element has a cross-section typical for a certain version of support.
- Element marked with blue joins outside wall of supporting shell with its axial line. This element is a rigid link.

Node marked with yellow is fixed per 5 degrees of freedom for fixed support (Fx, Fy, Fz, My, Mz) or 4 degrees for sliding support (Fx, Fy, My, Mz). In the process of solving, the fastening in Fz for the sliding support is iteratively modeled by the friction force.

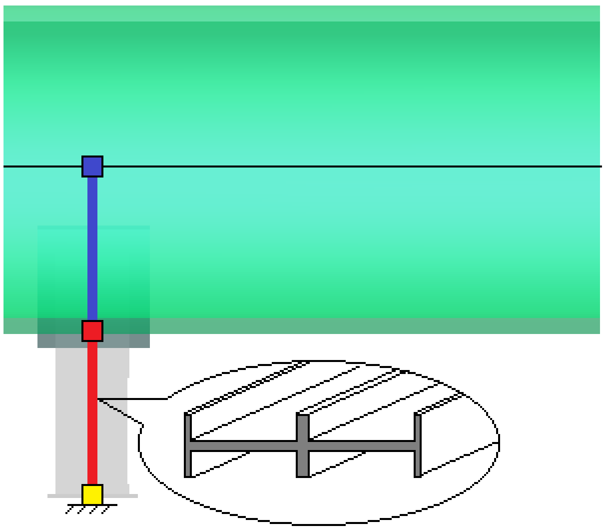


Fig. 3.98 Modeling a saddle support with beam elements

If analysis of the support is required (☑ Support calculation required), support type (one of several standard types), materials and dimensions must be assigned.

“Without calculation” option allows to form a refined version of the support without calculating (only the shell at the point of support is considered).

“Flip horizontally” option flips the support relative to the vertical plane.

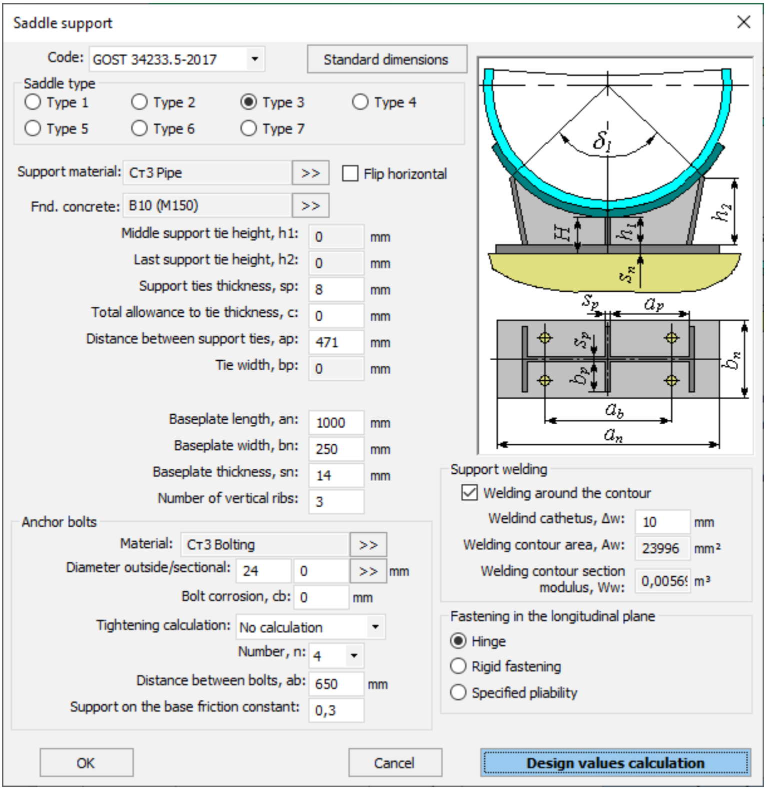


Fig. 3.99 Saddle support options

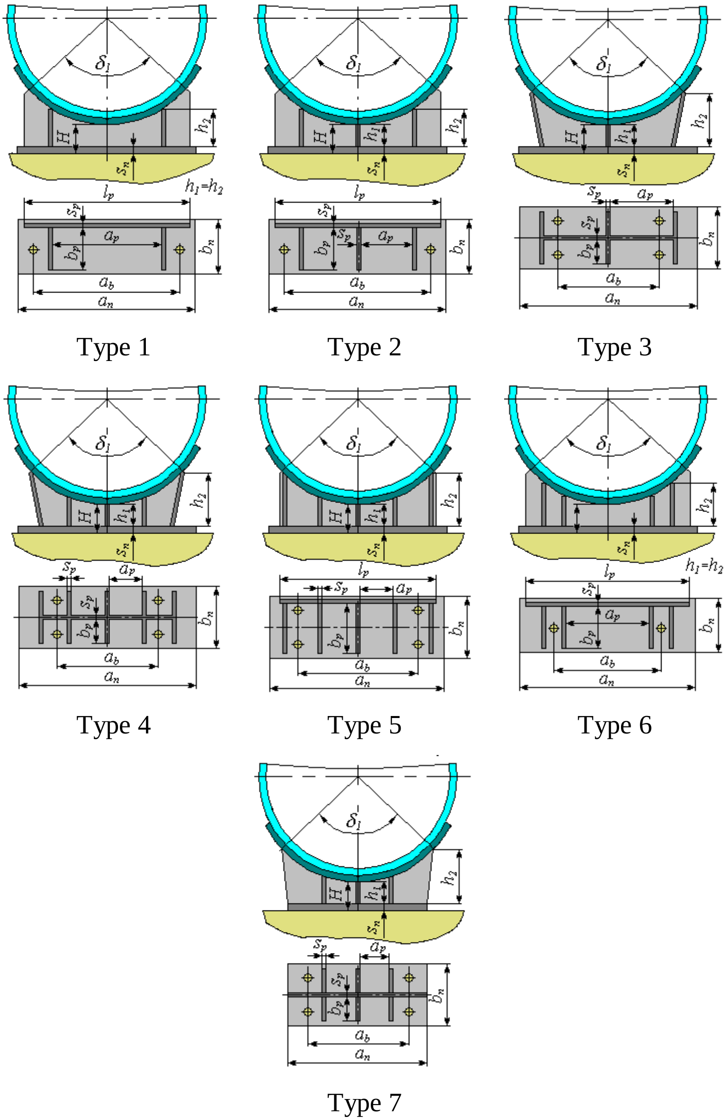


Fig. 3.100 Saddle types

The “Anchor bolts” option allows you to set and calculate the fastening of the support to the foundation under external loads (weight, seismic, wind, temperature loads).

The “Support welding” option allows you to set the parameters of the weld between the vertical ribs of the support and the base plate.

The option “Fastening in the longitudinal plane” allows you to control the fastening of the support from the moment in the YZ plane:

- Hinge – used for flexible structures or sliding supports if there is a gap between the base plate and anchor bolts, or for fixed supports if the anchor bolts are in a row in the XY plane. Fastening in the YZ plane is modeled by a hinge, no moment occurs in the reactions. This option is recommended by GOST 34233.5 (Fig. 3.101)
- Rigid fastening – used for rigid structures or fixed supports rigidly fixed to the foundation. Fastening in the YZ plane is modeled by a rigid anchorage, the shell body is modeled by a rigid element. This option gives the most conservative result, possibly a significant overestimation of the loads.
- Specified pliability – used in cases where it is possible to estimate the overall pliability of the fixing (flexibility of the shell wall + pliability of the foundation). Fastening in the YZ plane is modeled by a spring with a given pliability.

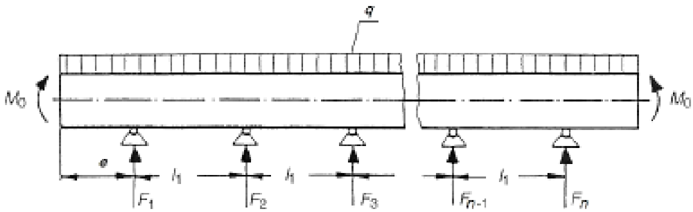


Fig. 3.101 Calculation scheme of a horizontal vessel according to GOST 34233.5

3.17.21. Bracket supports of horizontal vessel

This component is a group of two symmetrical supports. It can be attached to the same components of a horizontal vessel as the saddle support.

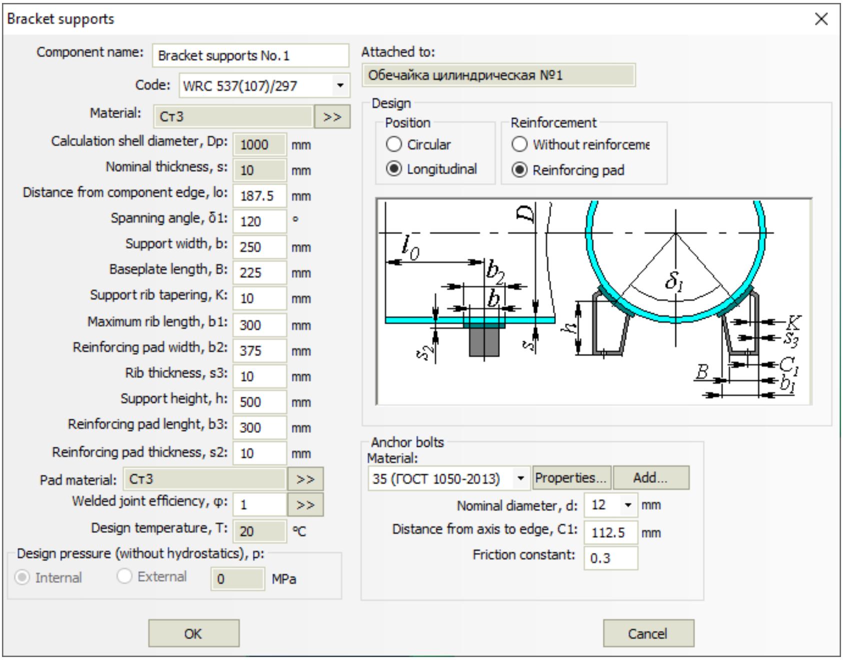


Fig. 3.102 Bracket supports of horizontal vessel

During calculation, the load on each support is determined individually, upon which the supporting shell is calculated on the impact of the local load applied along the welding contour of the support.

3.17.22. Bracket supports of vertical vessel

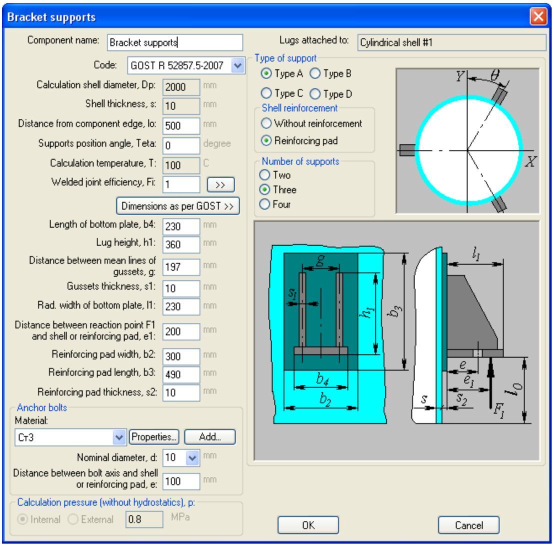


Fig. 3.103 Bracket supports of vertical vessel

Bracket support type is determined according to GOST 34233.5-2017 (see Fig. 3.104).

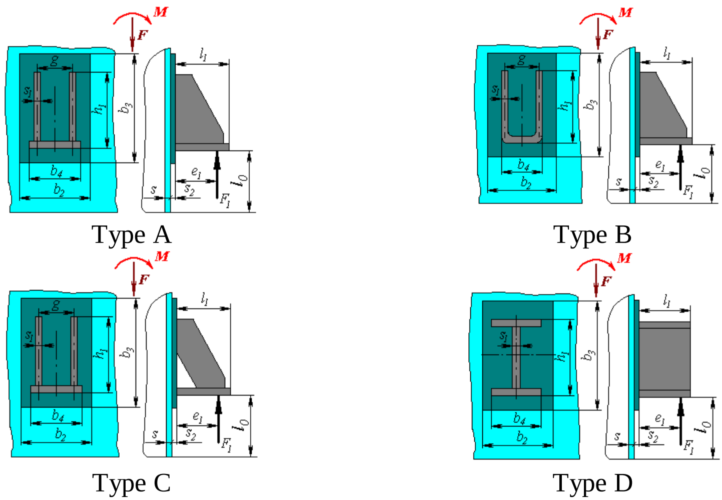


Fig. 3.104 Bracket supports types

Bracket supports can be adjoined to any cylindrical or conical shell or steep conical head of the vessel casing. Bracket support placement and dimensions determine the analysis of bearing loads on vessel components. Analysis is carried out if 2, 3 or 4 supports are present. If there are 4 supporting lugs, assembly quality must be accurately defined ☑ Accurate mounting. Name, code of standards and dimensions of bracket supports are set in the same way as those for cylindrical shells. Bracket supports can have no reinforcements or be supported by reinforcing pads.

Key "**Standard dimensions**" activates selection of typical support as per the conditional load, because, according to the applicable codes, not the support itself is calculated, but a vessel wall in the place of its connection.

3.17.23. Supporting legs

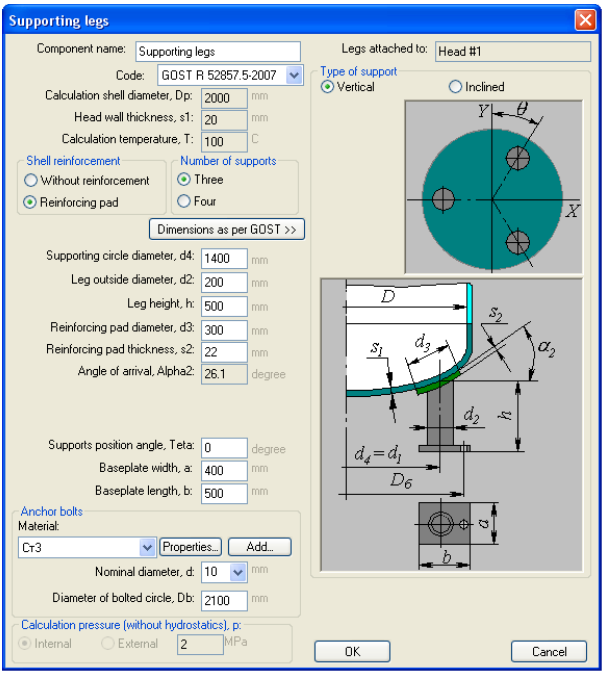
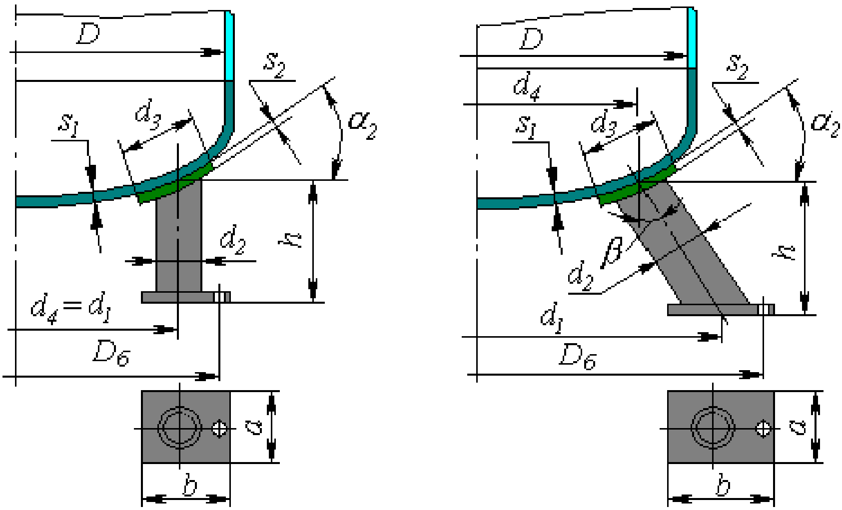


Fig. 3.105 Supporting legs

Supporting legs can be adjoined to the lower head of the vessel casing. Their placement and dimensions determine the analysis of bearing loads on vessel components. Analysis is carried out if 3 or 4 supporting legs are present. If there are 4 supporting legs, assembly quality must be accurately defined Accurate mounting.

Name and dimensions of supporting legs are set in the same way as those for cylindrical shells. Supporting legs can have no reinforcements or be supported by reinforcing pads (see Fig. 3.106).

Supporting leg type is determined according to GOST 26202-84 (see Fig. 3.106).



Type A Type B

Fig. 3.106 Supporting legs types

3.17.24. Supporting lugs

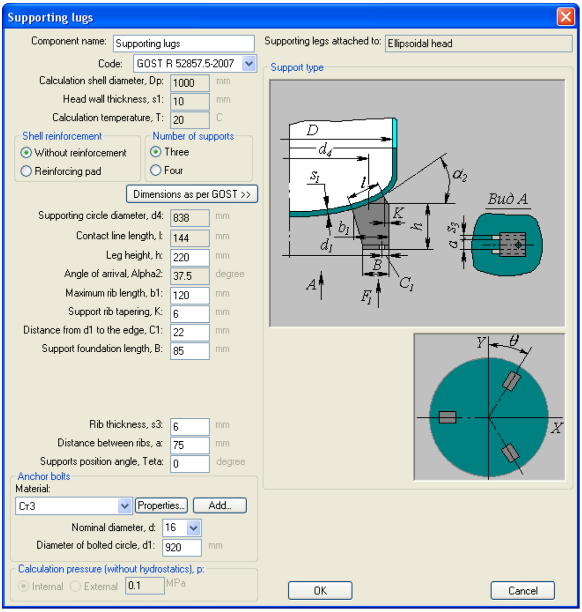


Fig. 3.107 Supporting lugs

Head lugs can be adjoined to the lower head, conical shell or steep conical head of the vessel casing. Their placement and dimensions determine the analysis of bearing loads on vessel components. Analysis is carried out if 3 or 4 head lugs are present. If there are 4 head lugs, assembly quality must be accurately defined ☑ Accurate mounting .

3.17.25. Supporting legs on the shell

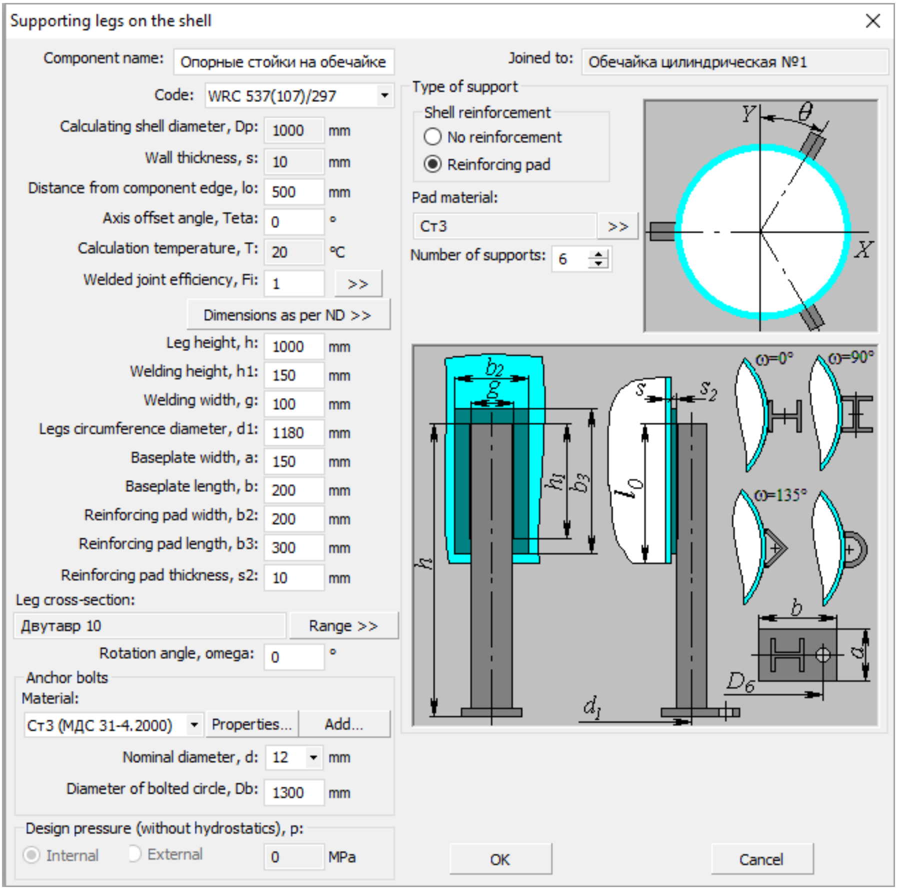


Fig. 3.108 Supporting legs on the shell

Supporting legs can be connected to the lower head or cylindrical shell of vessel casing. There can be any number of legs (not less than 2). Loads in the weld point of each leg are defined automatically from the analysis of statically undeterminable beam system, and are individual for each leg.

Note: The strength and buckling of the legs structure is checked simplistic, like a bar loaded with axial force.

3.17.26. Supporting ring

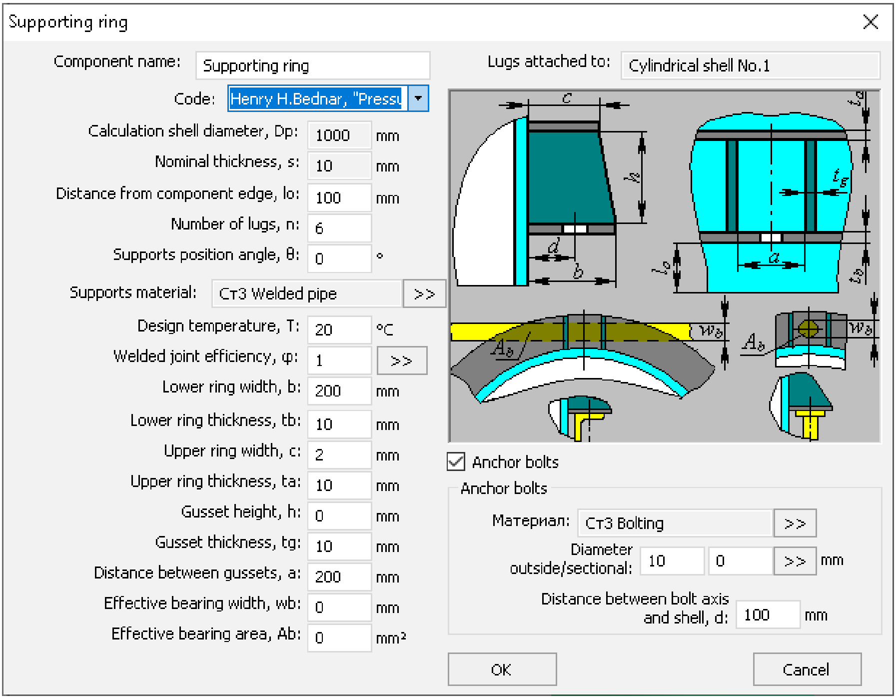


Fig. 3.109 Supporting ring as per H.Bednar

A ring support can be connected to the cylindrical shell of the vessel casing. The support is calculated as per Henry H. Bednar, "Pressure Vessel Design Handbook" [72].

A calculation according to EN 13335-3 [57], is also available, in which case the support configuration corresponds to Fig. 3.110.

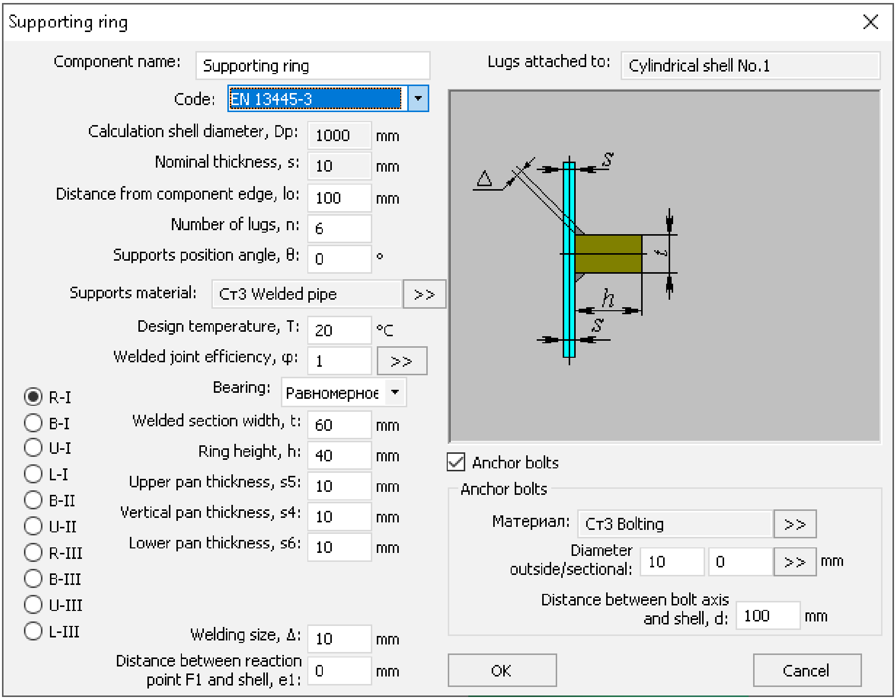


Fig. 3.110 Supporting ring as per EN 13445-3

3.17.27. Lifting lugs

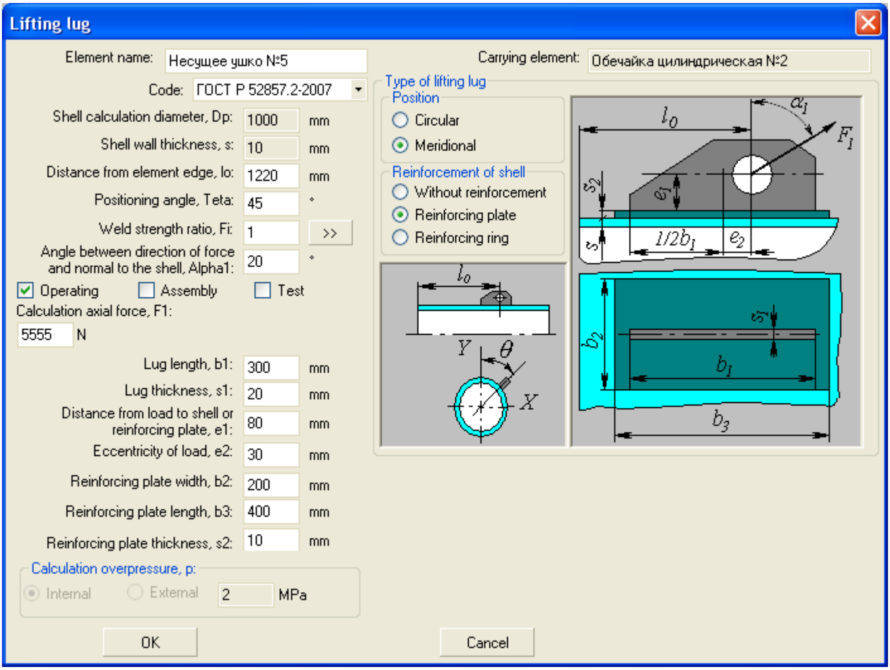
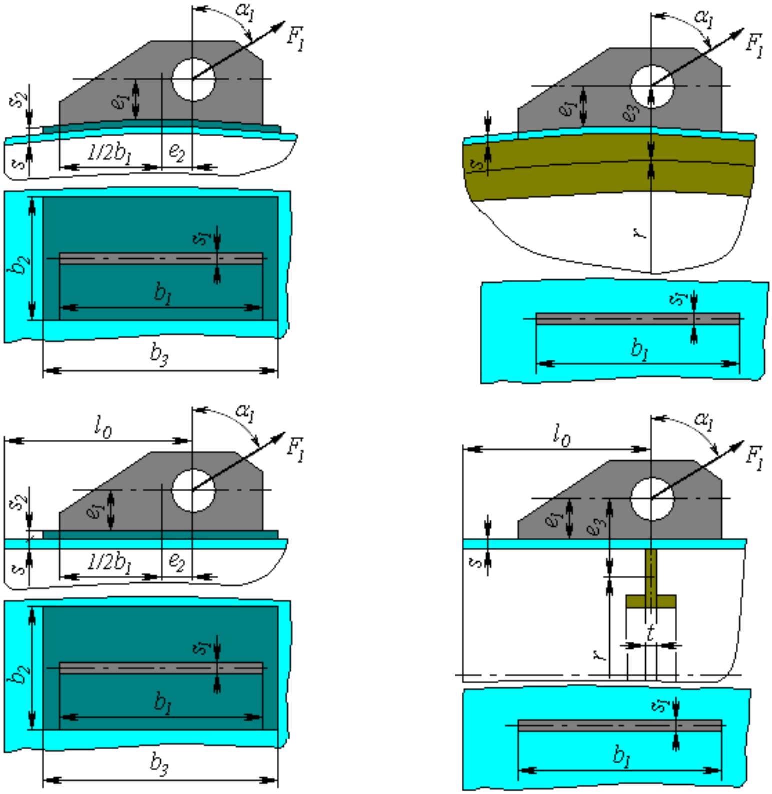


Fig. 3.111 Lifting lugs

Lifting lug can be adjoined to any cylindrical or conical shell of vessel casing, or to ellipsoidal (hemispherical) head. Loads and their direction must be input by user based on operation conditions. Lifting lugs can be placed both in longitudinal and transverse directions on the shell.



Lifting lug with reinforcing plate   Lifting lug with reinforcing ring

Fig. 3.112 Lifting lugs types

Lifting lugs can be either without reinforcement or reinforced by reinforcing plate or ring.

When reinforced by ring, its type, placement and dimensions are set in the same way as those for stiffening rings of cylindrical shells (i.3.17.18).The ring is considered when analyzing pressure influence on the cylindrical shell.

3.17.28. Joining pad

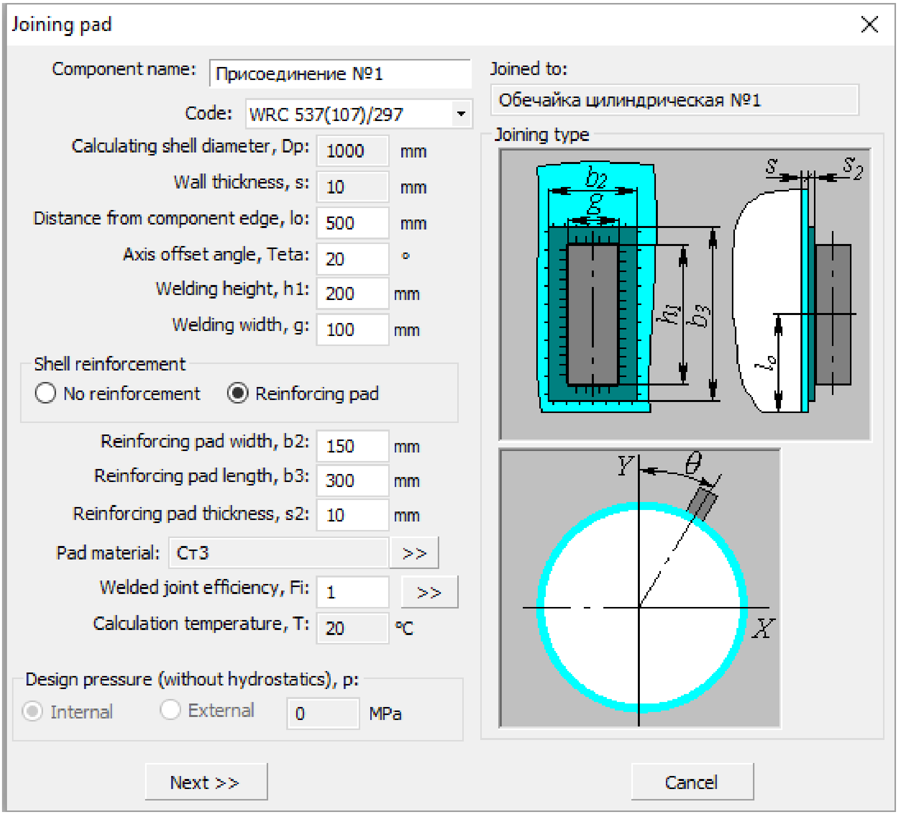


Fig. 3.113 Joining pad

This component is designed for modeling of any joints of external steel structures, consoles, non-standard supports of horizontal vessels, etc, with further calculation of carrying ability of casing wall as per WRC 537(107). Component can be joined to the cylindrical shell or spherical head. Loads for joining are set similar to component «Nozzle», and can be transferred to neighboring components of vessel and its supports.

3.17.29. Trunnion

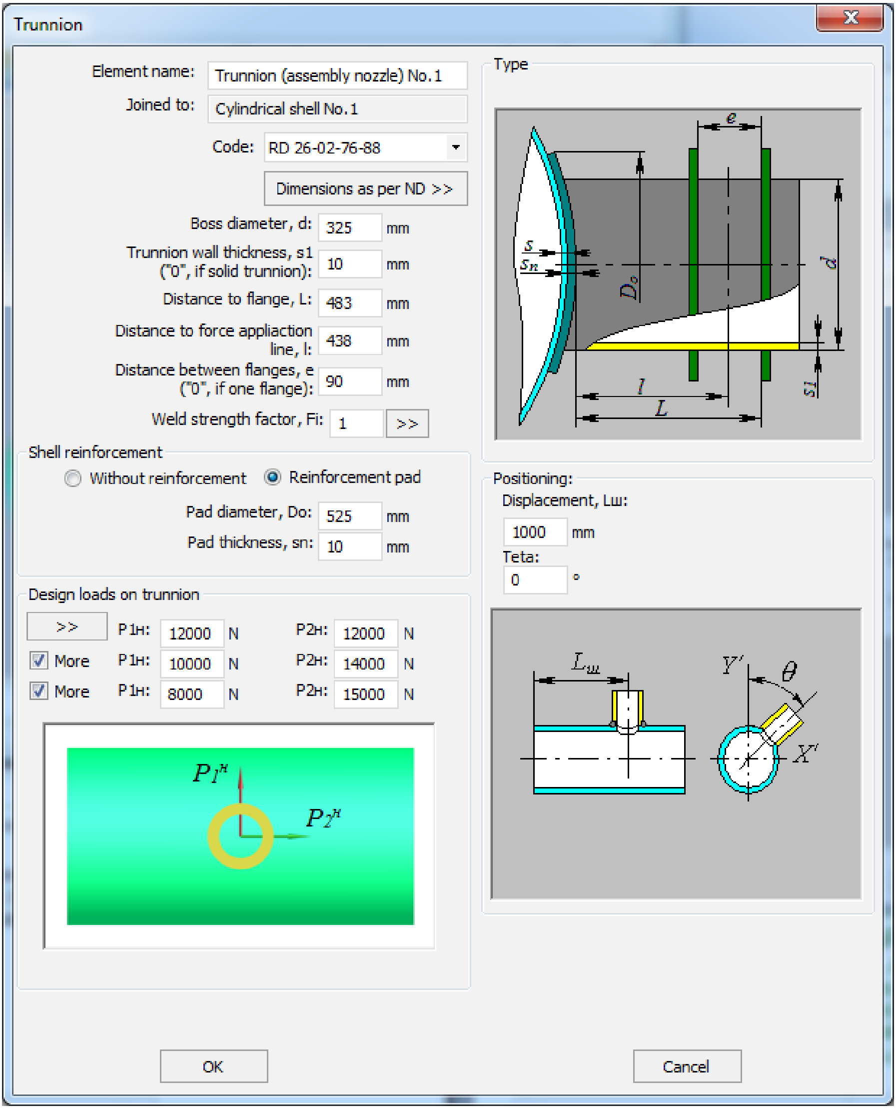


Fig. 3.114 Trunnion

This component can be joined to cylindrical shell. There are different variants of developing of this structure. If a solid boss is used, s_1 = 0 should be defined. Besides, there are variants with one or two stop flanges. If there is one flange, e = 0 should be defined.

There is a possibility to define up to 3 trunnion loading cases, with consideration for changing of loads at lifting. Pressing >> button automatically defines loads on the trunnion, if vessel weight, centroid position and point of lift are known:

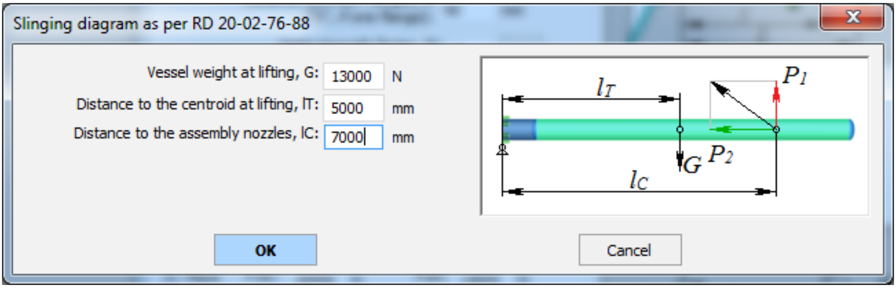


Fig. 3.115 Slinging scheme

For this component, export to Nozzle-FEM program is provided.

3.17.30. Additional loads

Besides loads from weight of shells, heads, fittings, etc., additional weight loads (for example, from service platforms) and force loads (for example, from adjoining pipes) can be input. **Fig. 3.117** includes an example of setting additional weight and external loads for horizontal vessels.

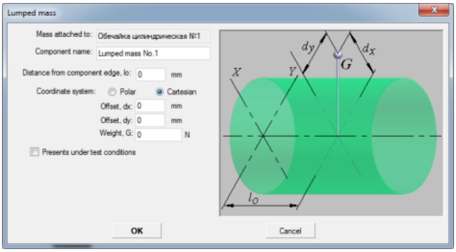


Fig. 3.116 Weight loads for horizontal vessels

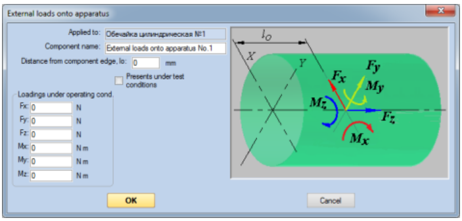


Fig. 3.117 External loads for vertical vessels

Lumped mass displacement is available, after which displacement moment will be calculated automatically.

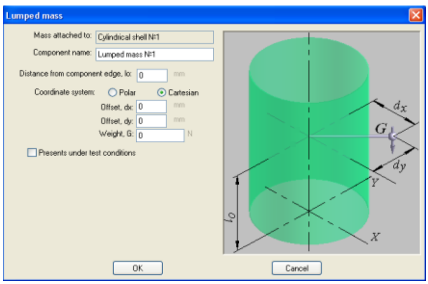


Fig. 3.118 Weight loads for vertical vessels

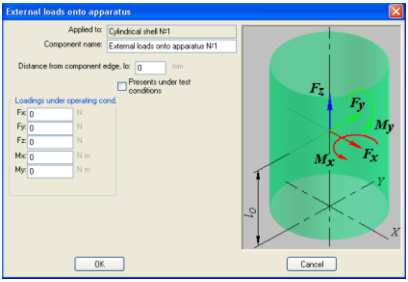


Fig. 3.119 External loads for vertical vessels

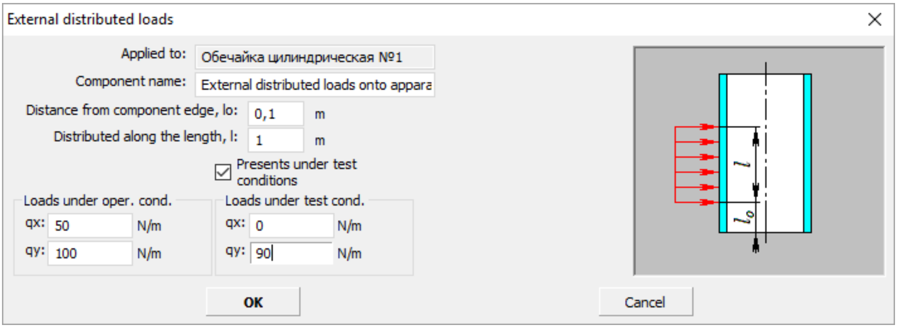


Fig. 3.120 External loads distributed along the component

3.17.31. Vessel fixing

This component is designed for consideration of non-standard fixing of vessel (that cannot be modeled via supports). These can be non-standard supports of horizontal vessels (legs or pillars), rigid steel structure enveloping the vessel (racks, apertures), as well as multilayer vessels (fixed lower nozzles act as supports).

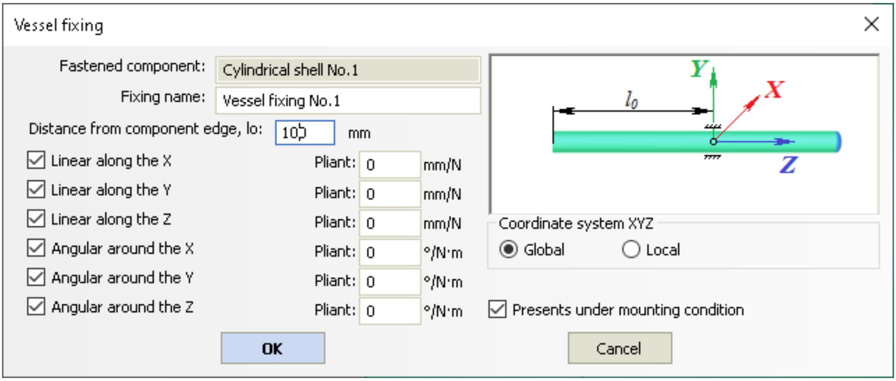


Fig. 3.121 Vessel fixing

It is necessary to specify fixed degrees of freedom in a global or local system of coordinates (the local system corresponds to the parent component's coordinate system). Results of calculation will be appropriate reactions of supports.

To simulate rigid fixing by the corresponding degree of freedom, it is necessary to set the pliant equal to 0.

3.17.32. Service platform

The platform can be installed on the cylindrical parts of vessel casing, as well as on the supporting skirt of the column.

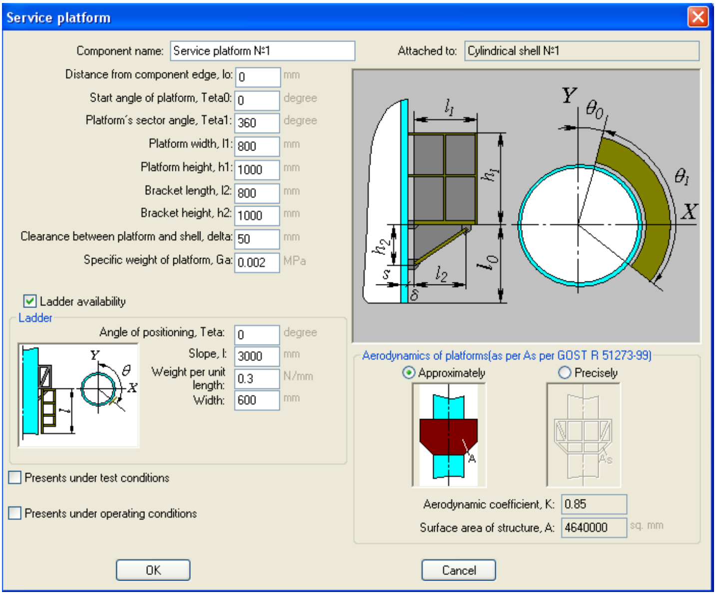


Fig. 3.122 Service platform of vertical vessel

A variant of the platform for the horizontal vessel is a rectangular flooring, optionally fenced with railings. Weight of the platform and its wind load are applied to the horizontal shell in a given number of points (parameter “Number of rows of supporting lugs”).

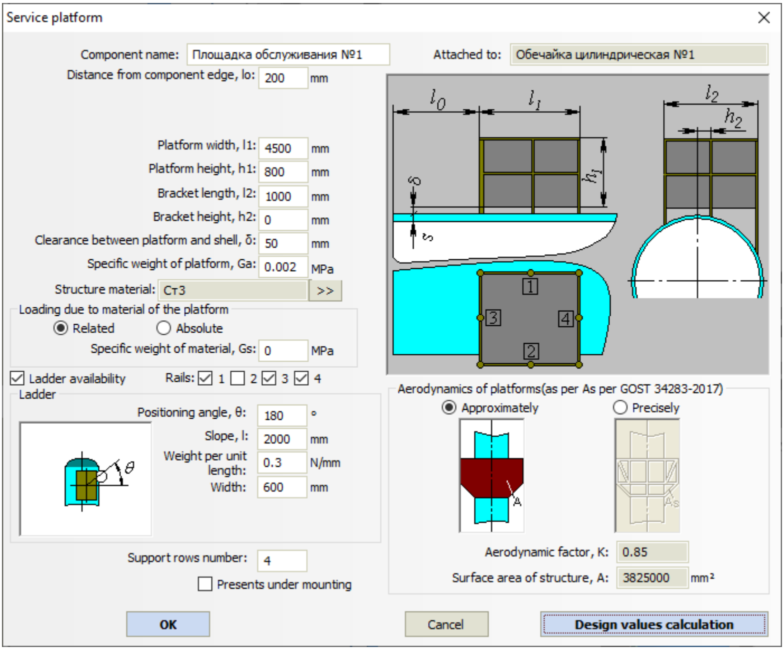


Fig. 3.123 Service platform of horizontal vessel

Combination of railings on the four sides can be arbitrary, which makes it possible to form a multi-layer floor from several platforms.

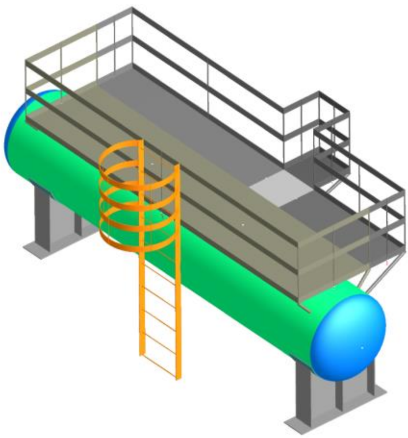


Fig. 3.124 Service platforms group

The specific weight G_a is assigned according to SP 20.13330 [35] and includes the load from the material, snow, equipment, people, etc. This load is considered to be evenly distributed over the sector of the site and is used in calculations (when determining the natural period, load on the supporting shell, etc.).

The parameter "Weight of material" is not used in strength calculations and is required only for calculating material consumption.

3.17.33. Column components

Packings, service platforms, trays, concentrated masses and external loads can be adjoined to cylindrical shells of columns. Their position, dimensions and mass (forces) are added to loads and considered in vessel strength and stability analysis under wind and seismic loads.

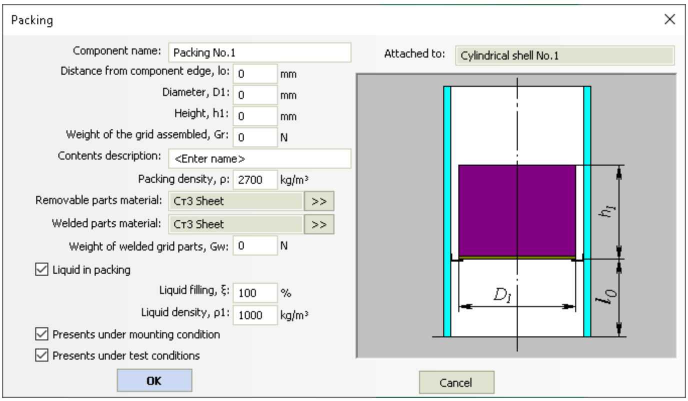


Fig. 3.125 Packing

When calculating the weight load, the packing is considered as a complex component containing:

- welded part weighing G_w (taken into account in any design mode);
- removable part weighing G_r-G_w (taken into account in the design mode according to the options "Present under mounting/test conditions");
- liquid (if any) in the form of a conditional cylinder weighing $\rho_1 \cdot \pi \cdot D_1^2/4 \cdot h_1 \cdot \xi_1$ (taken into account in operating conditions);
- filler (catalyst) in the form of a conventional cylinder weighing $\rho \cdot \pi \cdot D_1^2/4 \cdot h_1$ (always taken into account in operating conditions, in the mounting/test conditions according to the specified options)..

The table of materials used is formed taking into account the materials assigned to the welded and removable part..

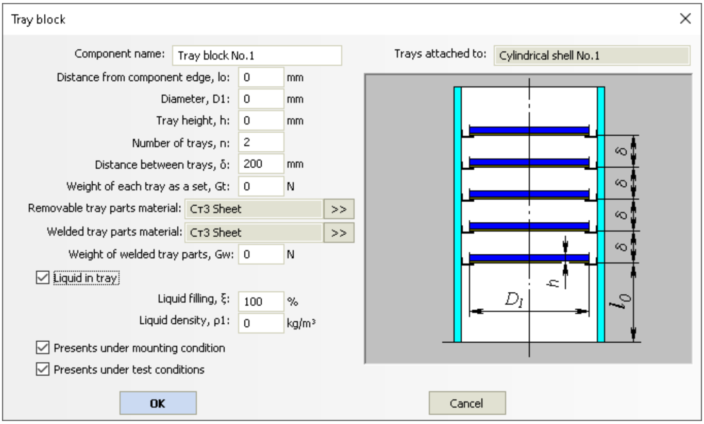


Fig. 3.126 Tray block

When calculating, a group of trays is considered as several lumped masses attached to a parent component with an equal pitch. The weight load of each tray is taken into account in the same way as for the “Packing” component.

When rendering the model, the trays are displayed conditionally. If necessary, you can combine them with the "Custom Equipment" component, preparing a tray of any required design in any CAD system.

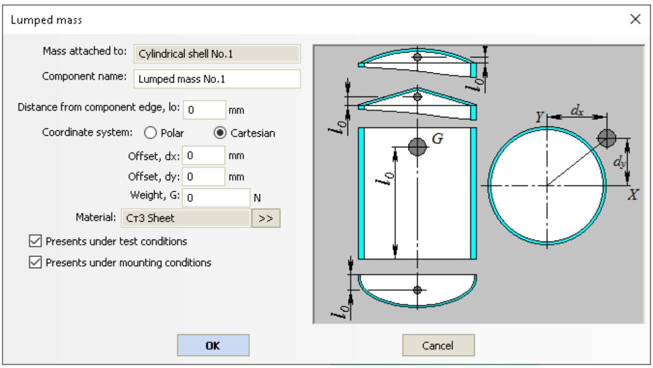


Fig. 3.127 Lumped mass

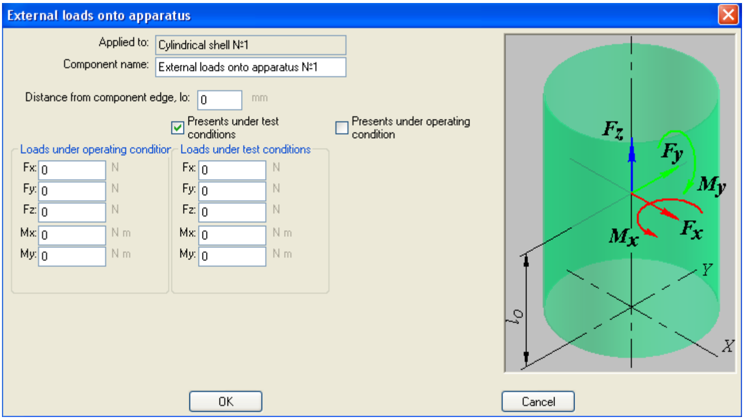


Fig. 3.128 External loads

3.17.34. Skirt support

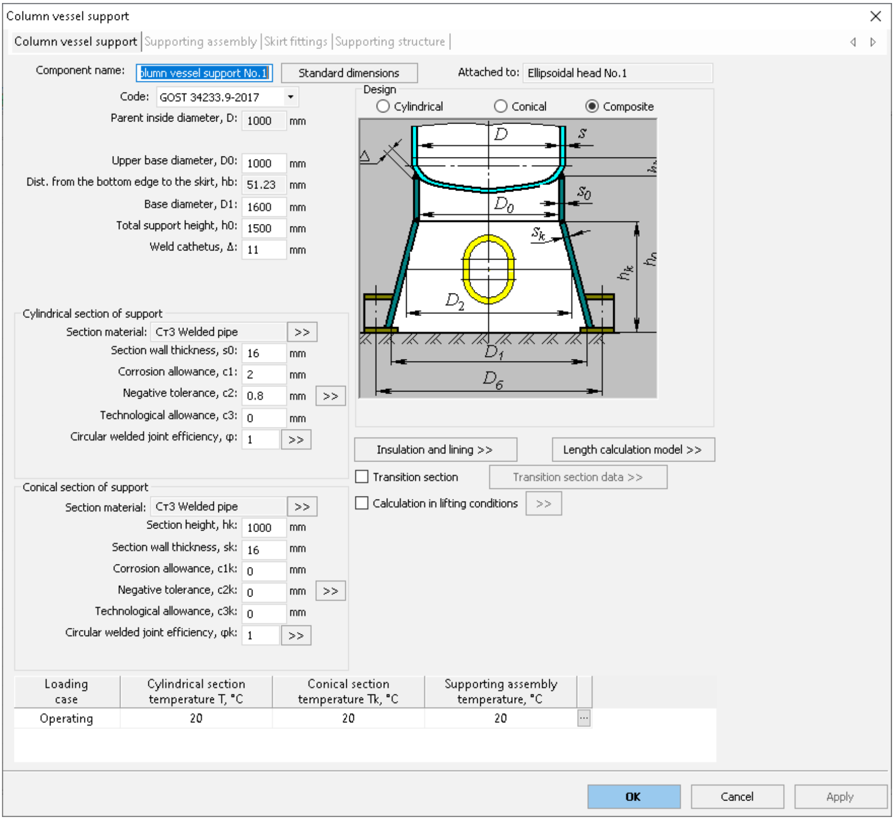
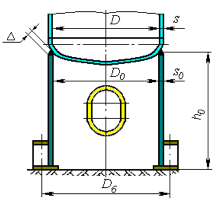
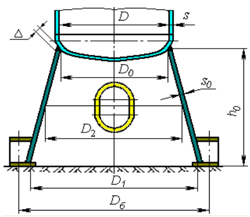


Fig. 3.129 Skirt support

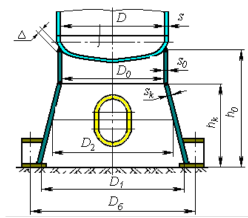
Skirt support can be adjoined to the model's lower head or to the cylindrical or conical casing shell. Support shell type is determined according to GOST 51274-99 (see Fig. 3.130).



Cylindrical



Conical



Composite

Fig. 3.130 Skirt types

There is a possibility of express temperatures estimation for the support components. To do this, click the […] button in the table of loading cases. In the dialog that appears, you can select a method for the temperatures estimation (Fig. 3.131).

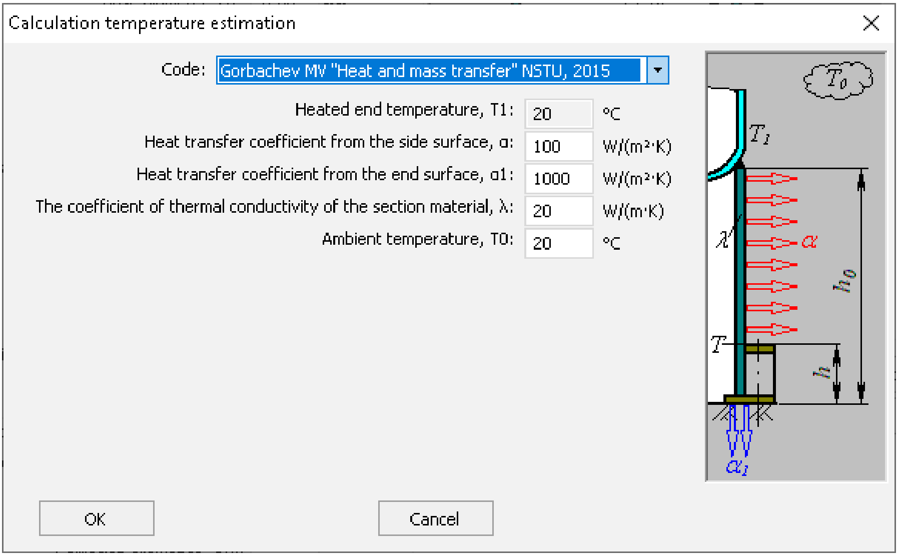


Fig. 3.131 Skirt components temperature estimation

A transitional section (skirt) with a material different from the support material can be input. To set the transitional section's dimensions and material, select ☑ Transition section and press Transition section data >>.

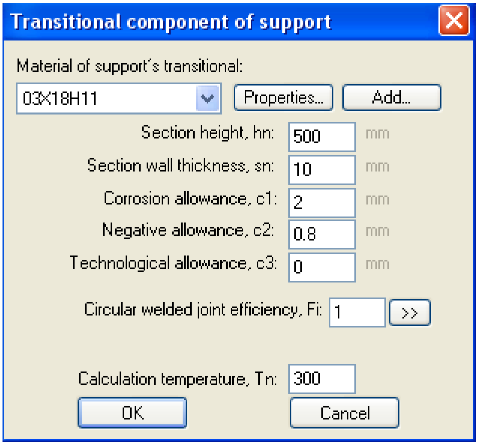


Fig. 3.132 Transitional section

Option ☑ Calculation in lifting conditions enables performing of supporting shell calculation of strength and stability against loads arising during mounting of the column, and selecting of additional furnishings. For this purpose, additional data should be defined (Fig. 3.133).

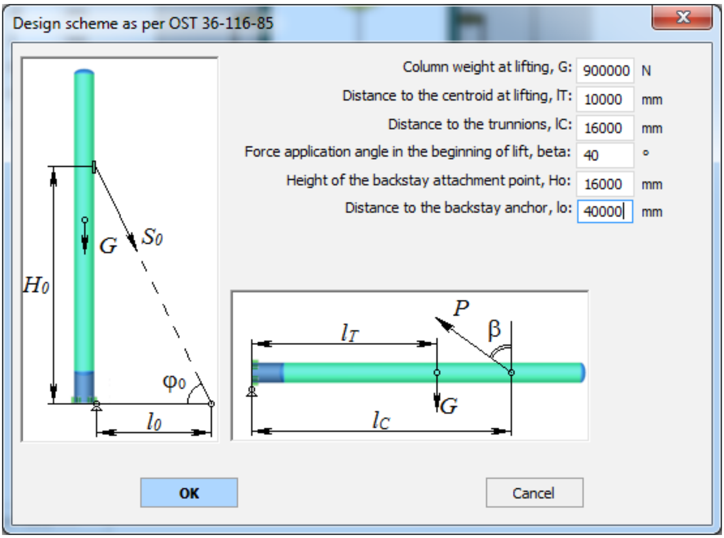


Fig. 3.133 Lifting scheme

Supporting assembly type and dimensions can be input by pressing Next >> .

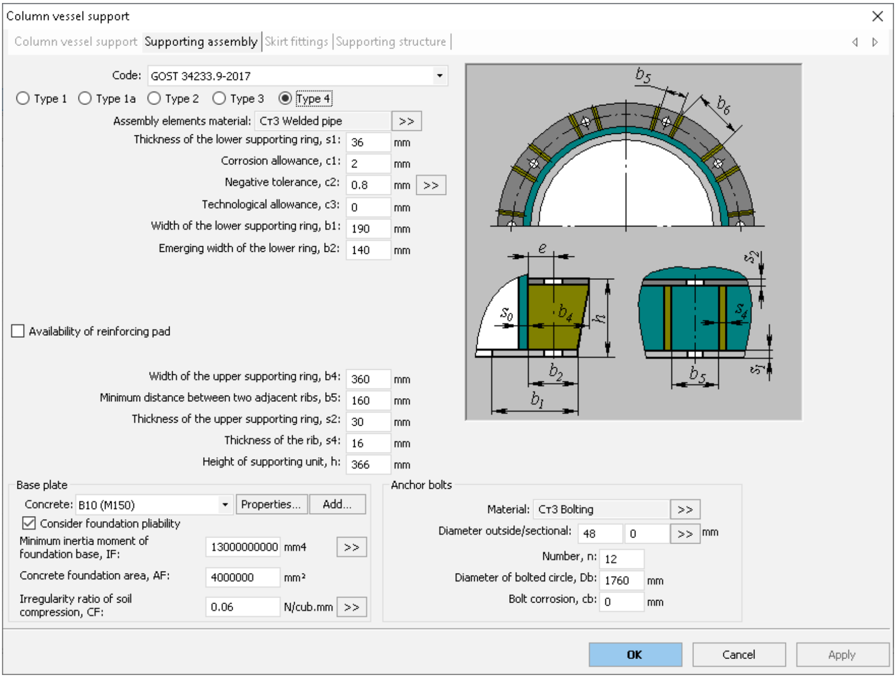
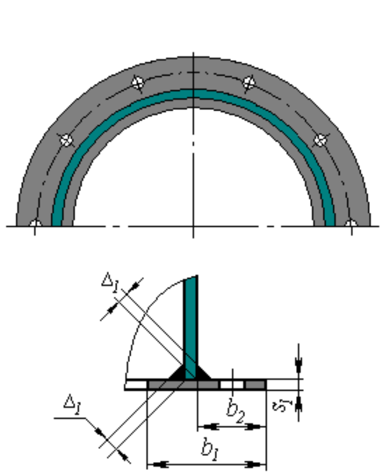
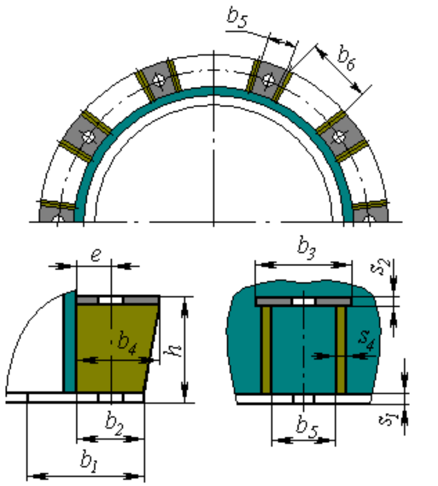


Fig. 3.134 Supporting assembly

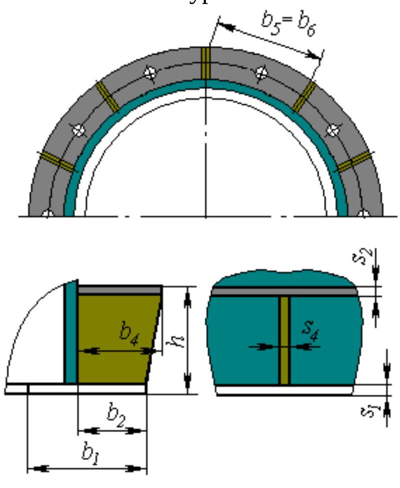
Supporting assembly type is determined according to GOST R 51274-99 and GOST 24757-81 (see Fig. 3.135).



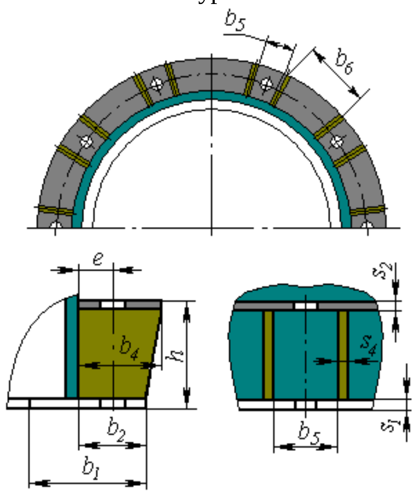
Type 1



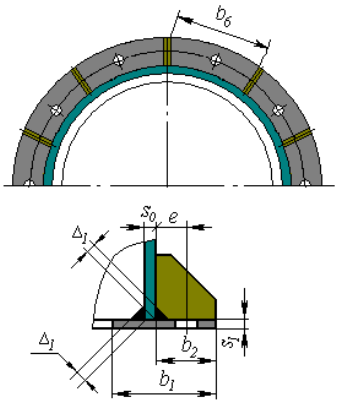
Type 2



Type 3



Type 4



Type 5

Fig. 3.135 Supporting assembly types

Option ☑ Consider foundation pliability enables considering influence of soil flexibility on natural vibrations period of column. At the active check, it is necessary to define area and inertia moment of foundation base, as well as irregularity coefficient of soil compression. To define geometric characteristics, there is an assistant button, which enables automatic calculation of typical foundation parameters.

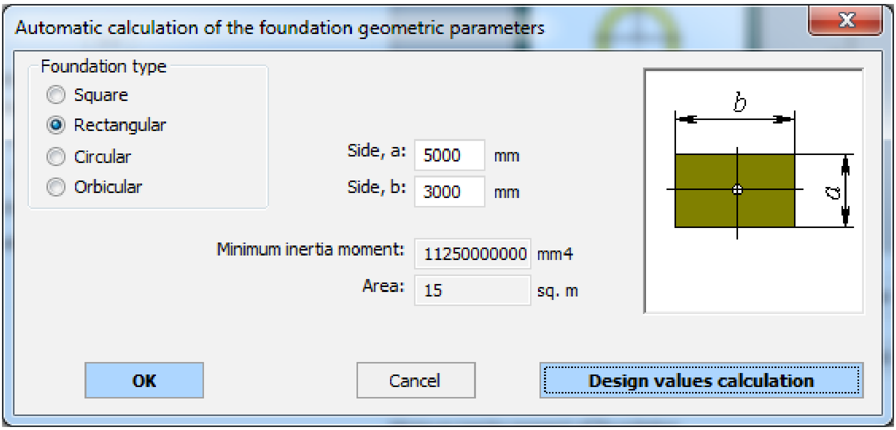


Fig. 3.136 Foundation geometric parameters

When the check is off, the foundation is modeled as built-in.

Supporting shell fittings can be input by pressing [Next >].

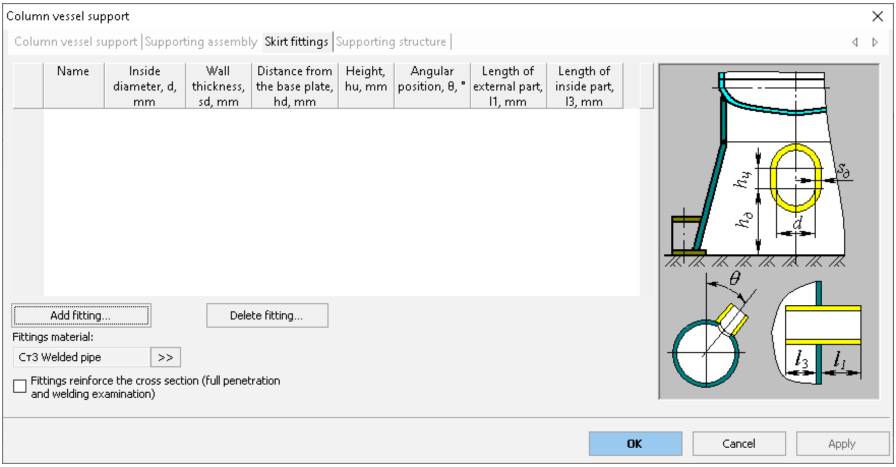


Fig. 3.137 Supporting shell fittings

Any number of fittings (including stretched), as well as their dimensions and placement, can be input using [Add...] [Copy...]. During supporting shell analysis, all sections will be checked and most unsafe section will be determined.

The option “Fittings reinforce the cross section” controls the way of defining the skirt cross section characteristics (when this option is enabled, the cross section is formed taking into account the walls of the fittings).

Support structure (pedestal), if present, can be input by pressing [Next >] (☑ Availability of supporting structure).

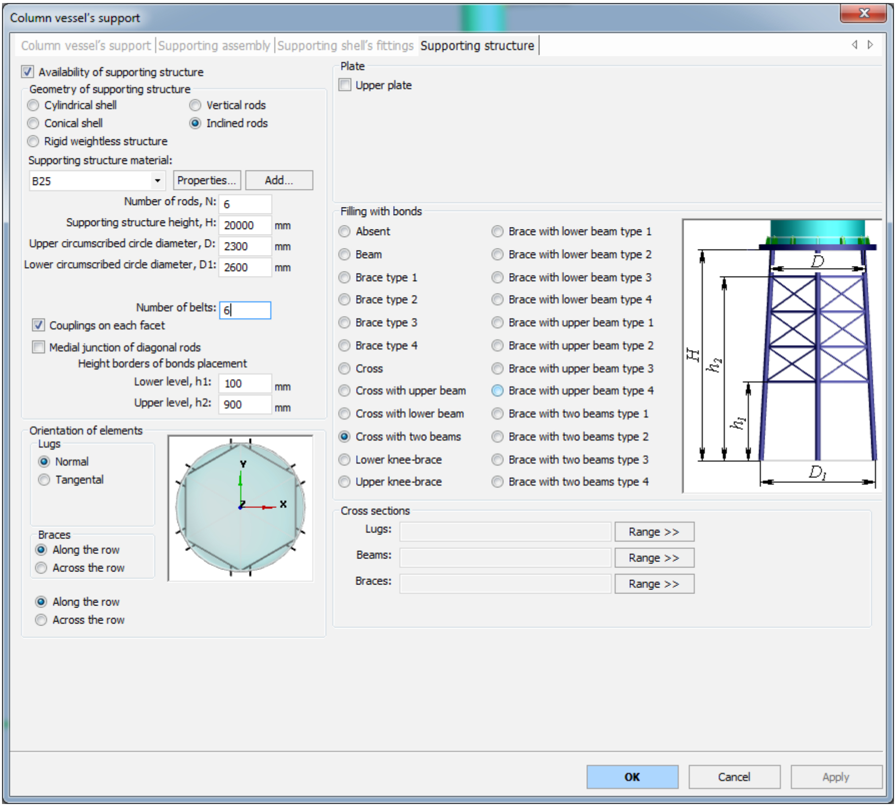


Fig. 3.138 Column support structure

The support structure can be in the form of cylindrical or conical shells or metal structure with a foundation of vertical or tilted poles.

Variant «Rigid weightless structure» is designed for cases, when pedestal parameters are not known yet, but its height is known. In this case, the pedestal is modeled by rigid link and causes no influence on the vibration period. Wind loads are calculated with consideration the pedestal height.

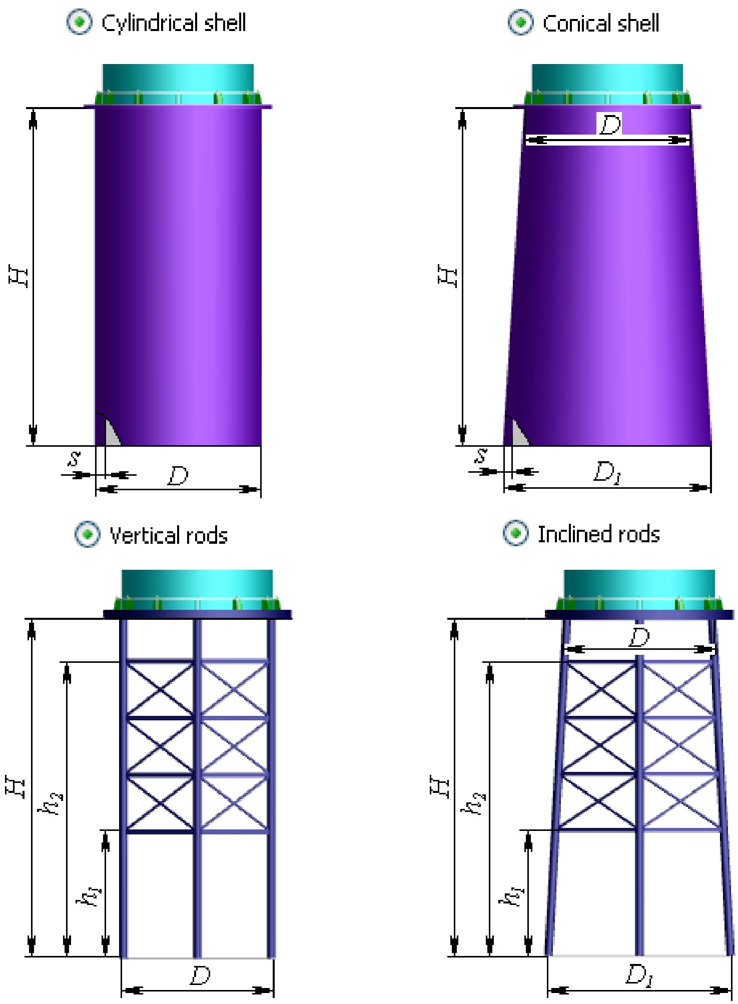


Fig. 3.139 Support structure types

3.17.35. Heat Exchanger with stationary tube plates

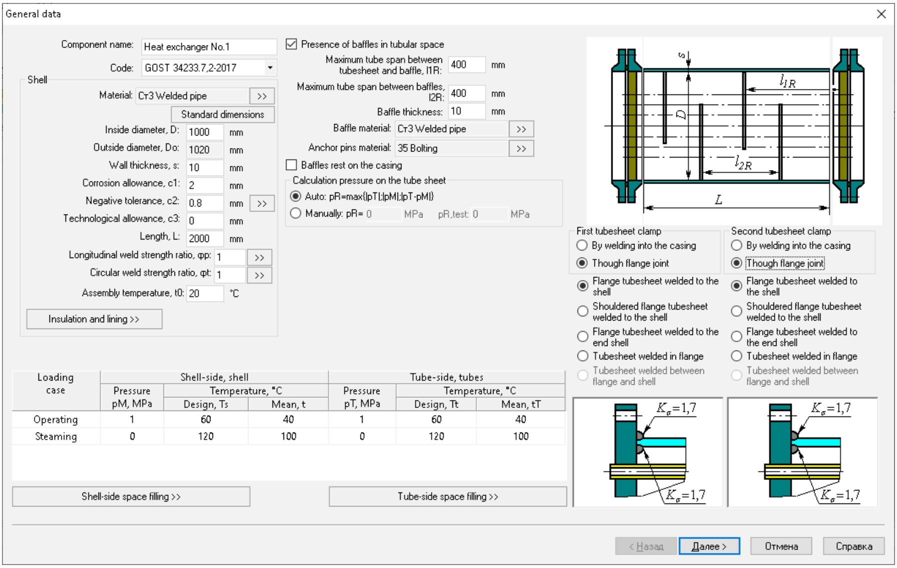


Fig. 3.140 Heat Exchanger Data

Heat exchanger casing wall temperature is used to determine allowable stress on the casing. Average casing wall temperature is used to determine the linear expansion and elasticity factors. Insulation and lining can also be input.

Operation environment properties in tubular space are determined by the heat exchanger's parent component or (if there is no parent component) using the "General Data" setting. Operation environment properties in inter-tubular space are considered when calculating weight of the casing's daughter components.

Casing and tube plate joint structure according to GOST 34233.7-2017 (RD 26-14-88) are shown in the Fig. 3.141.

A heat exchanger can be calculated according to ASME VIII-1. In this case, variants of structure are shown in the Fig. 3.142.

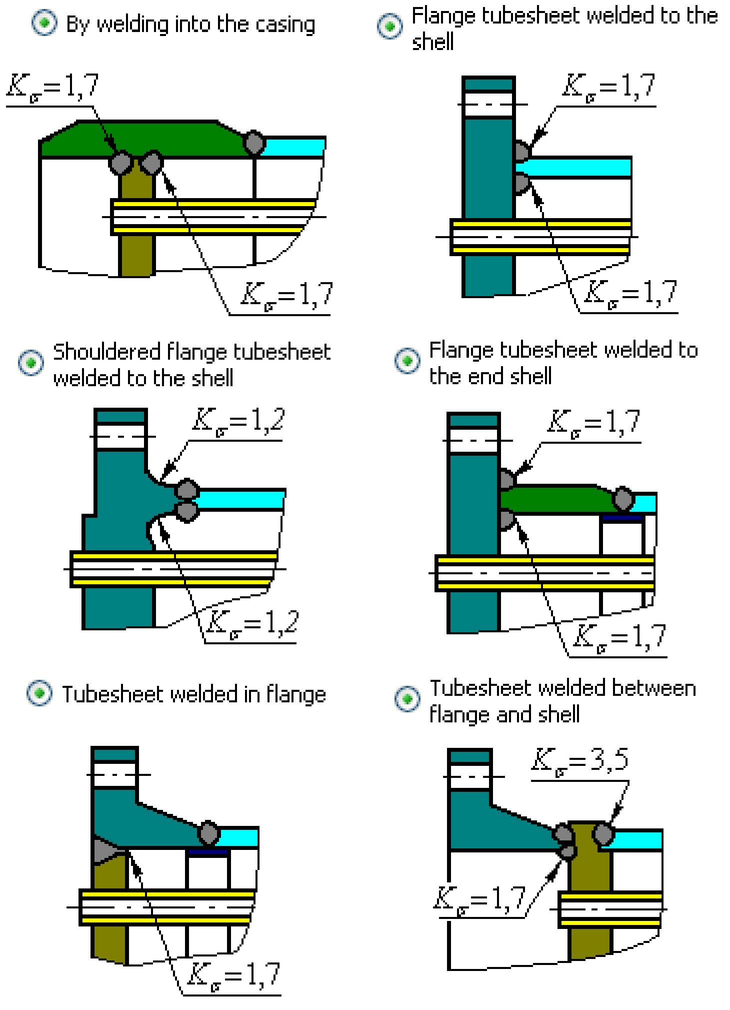
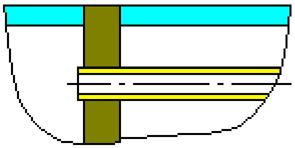
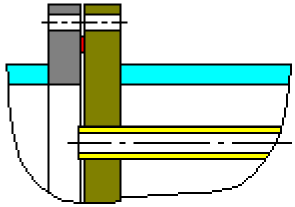


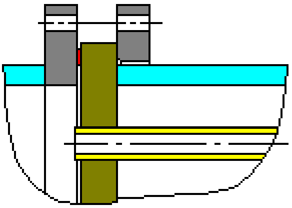
Fig. 3.141 Tube plate joints



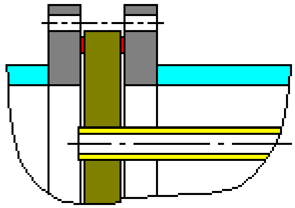
UHX-13.1 (a)



UHX-13.1 (b)



UHX-13.1 (c)



UHX-13.1 (d)

Fig. 3.142 Tube plate fixing as per ASME VIII-1

Properties of the tube plate joint can be input by pressing Next > (see i.3.17.36).

3.17.36. Tube plate joint

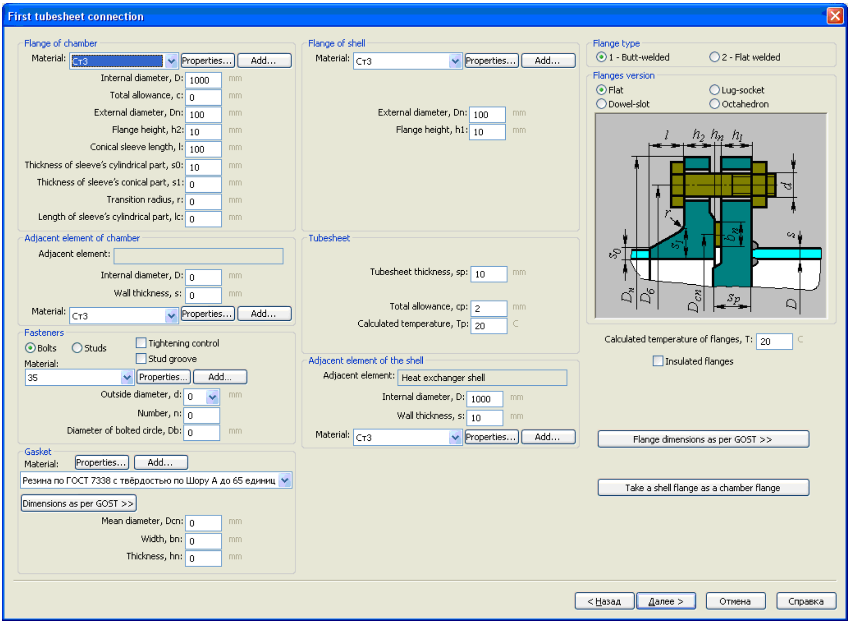


Fig. 3.143 First tube plate joint

If a tube plate is connected through a flange joint, data are input in the same way as those for a flange joint, according to GOST 34233.4-2017 (RD 26-15-88). Standard flanges can be selected.

Option "Transitional shell" is available for all variants of design of the tube sheet. When this option is activated, an additional window appears with parameters of the transitional shell (bushing), Fig. 3.124.

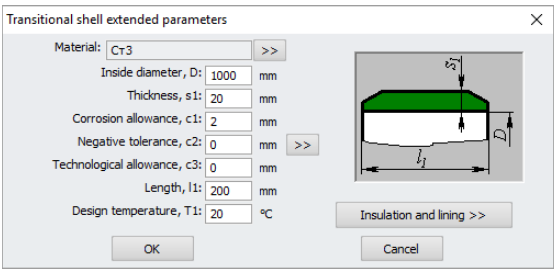


Fig. 3.144 Parameters of the transitional shell

Properties of the second tube plate joint can be input by pressing Next > . Data on the second tube plate is assigned equally to the first tube plate. You can quickly copy data from the first tube plate by pressing button "Accept the second connection as the first". Tube bundle properties, properties of tube mounting within the plate and pass partitions (if present) can be input by pressing Next > (see i.3.17.37).

3.17.37. Tube bundle properties

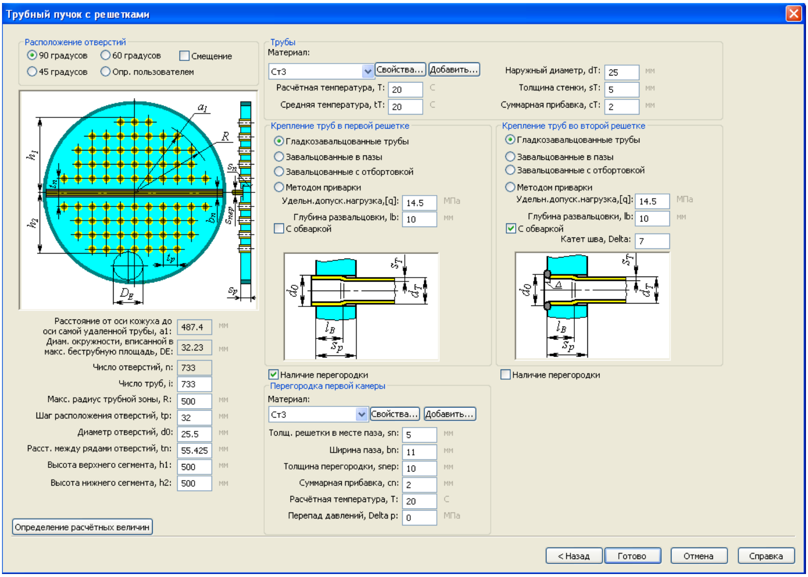


Fig. 3.145 Tubes bundle with plates

Position of passages in the tube plate can be set manually or automatically. To set position automatically, position angle of opening axis, opening spacing and diameter, radius of tubular space, and heights of upper and lower sections must be input. Properties such as the number of openings, distance to the most distant pipe's axis and the maximum diameter fitting within the tubeless area will be calculated automatically.

The position of symmetry axis of tube bundles can be changed by selecting ☑ Offset (Fig. 3.146)

◉ 90 degrees ◉ 60 degrees

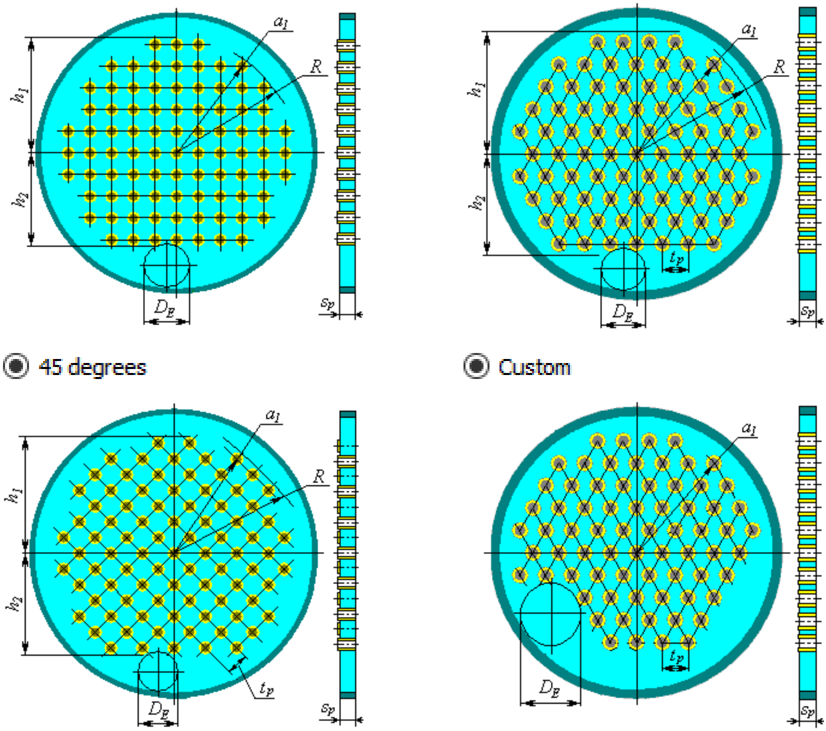


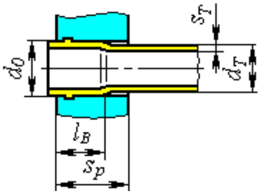
Fig. 3.146 Holes layout

At non-standard parameters of tube bundle, it is possible to compose it in the interactive designer mode.

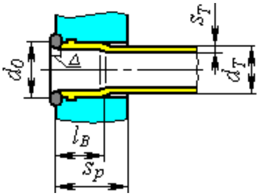
Constructions of tube holders in the sheet as per GOST 34233.7-2017 (RD 26-14-88) are shown on Fig. 3.147

Smoothly rolled

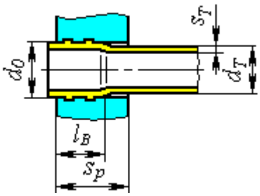
Smoothly rolled, with seal welding



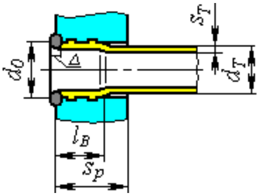
Rolled in one groove



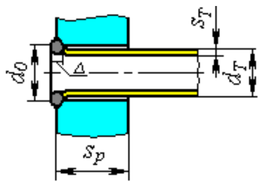
Rolled in one groove, with seal welding



Rolled in two and more grooves



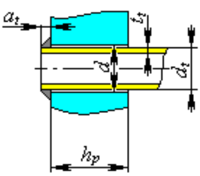
Rolled in two and more grooves, with seal welding,



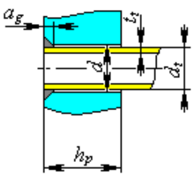
Welded without rolling

Fig. 3.147 Tube holders in the sheet as per GOST 34233.7-2017

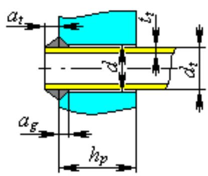
At calculation of the heat exchanger as per ASME VIII-1, possible types of holders are shown on Fig. 3.148.



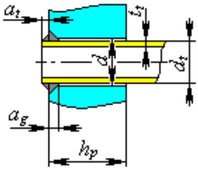
UW-20 (a)



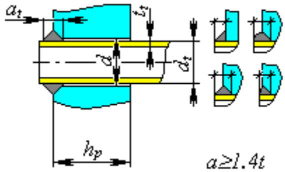
UW-20 (b)



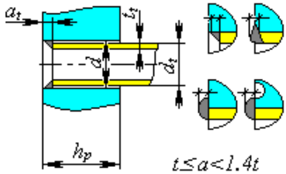
UW-20 (c)



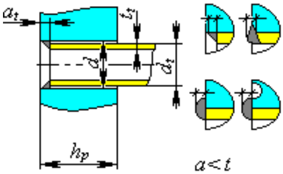
UW-20 (d)



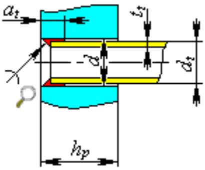
App.A (a)



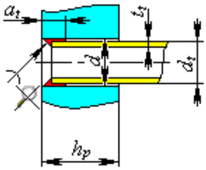
App.A (b)



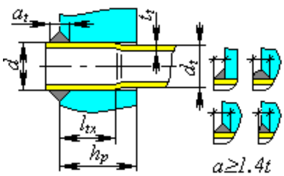
App.A (b-1)



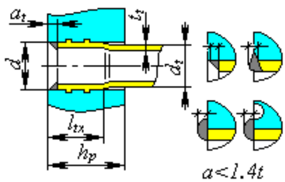
App.A (c)



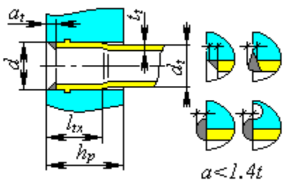
App.A (d)



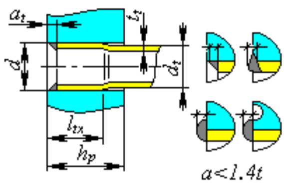
App.A (e)



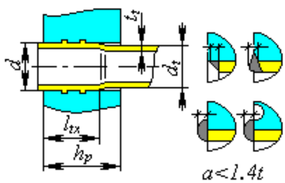
App.A (f)



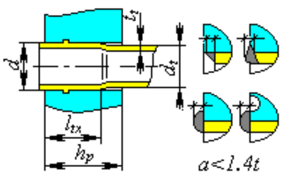
App.A (g)



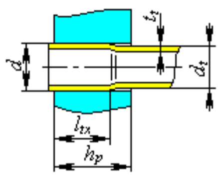
App.A (h)



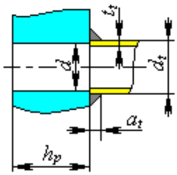
App.A (i)



App.A (j)



App.A (k)



UHX-11.1(d)

Fig. 3.148 Tube holders in the sheet as per ASME VIII-1

Optionally it is possible to add bellows (i.3.17.38) and (or) expander(s) (i.3.17.39) on the casing.

The heat exchanger will be displayed after all required data are input and saved by pressing Finish. Heat exchanger properties can be edited through a tabbed dialog box.

3.17.37.1 Handling with tube sheet designer

Designer enables creation of the tube bundle and calculation of its properties, when placing the tubes by method that is not described in the i. 3.17.37. To activate it, select the "Custom" option in the toolbar "Position of holes" A window specified in Fig. 3.149 will open.

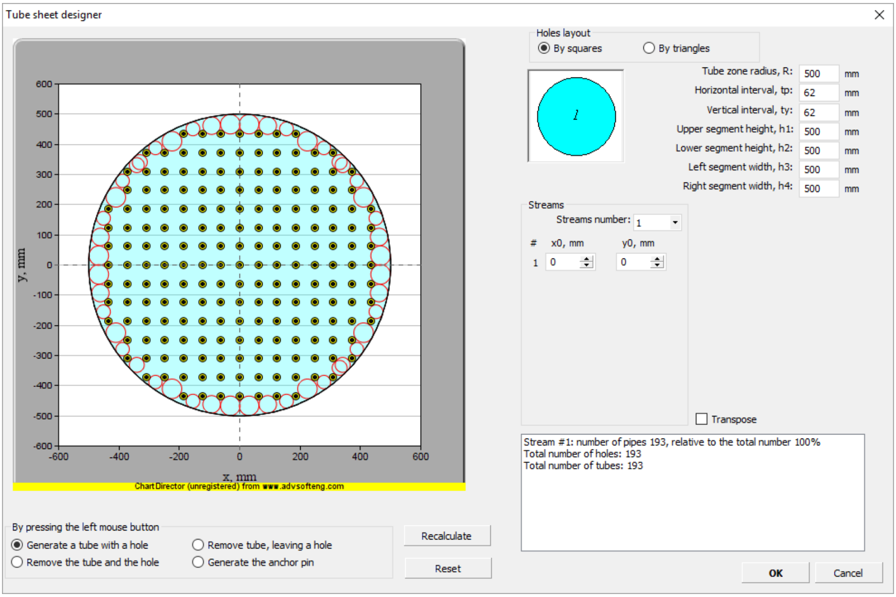
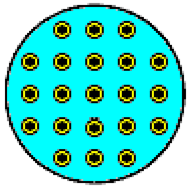
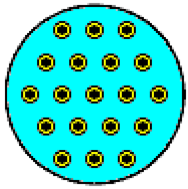
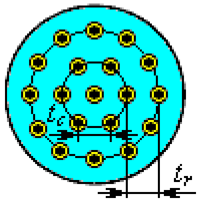
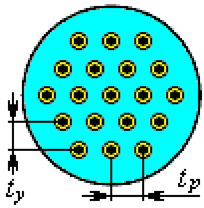
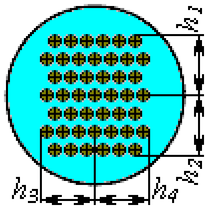


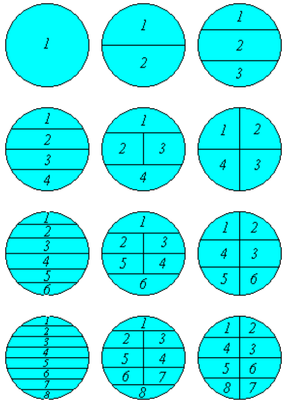
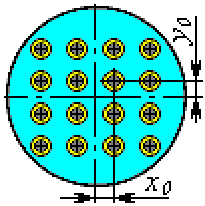
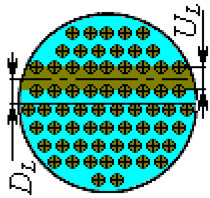
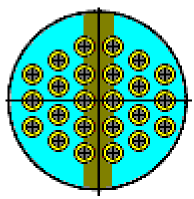
Fig. 3.149 Tube sheet designer

For window with the tube sketch, the following operations are available:

- Image scaling– by mouse wheel;
- Sketch moving – press and hold down left button of the mouse, and move it;

Commands and parameters:

Name	Description	Sketch
By squares	Rows of holes are not displaced	
By triangles	Each row of holes is shifted relative to the previous one for a half of the horizontal pitch	
By circles	Holes are arranged in concentric circles, holes on one circle are spaced with equal pitch, rounded to an integer number of holes	
t_p, t_y	Horizontal and vertical pitches of rows arrangement	
R, h_1, h_2, h_3, h_4	Enables creation of zone, beyond which arrangement of pipes is excluded	

Number of flows, configuration	Enables creation of pipeless zones for separating walls, for typical configurations of multiflow heat exchangers	
x_0, y_0	Provide more precise positioning of the tube bundle, belonging to the 1st flow, if the automatic arrangement does not furnish the desired result	
U_L	Distance between axis for pipes in the area of the 1st separating wall	
D_L	Distance from axis of tube sheet to axial line of 1st separating wall	
Transpose	90° rotation of the created tube sheet	
Create a tube with hole	If the tube already exists at the point with selected coordinates, than nothing happens, otherwise, the point gets an attribute "tube"	

Delete tube, remain hole	A hole will be included in calculation of the sheet peripheral zone, but the tube will be excluded from calculation of axial force on the casing.	
Delete tube and hole	There is no hole in this point, it cause no influence on calculation of peripheral zone (used for creation of tubeless zones)	
Create anchor stud	A hole will be included in calculation of the sheet peripheral zone, but the tube will be excluded from calculation of axial force on the casing	
Update	Rebuilding of tube bundle is performed (coordinates of points, where the holes will be placed, are outlined)	
Reset	All additional signs of points are cleared	

3.17.38. Heat Exchanger with expansion bellows on the casing

To include the expansion bellows in heat exchanger model, select an appropriate checkmark in the "Expansion bellows" tab (Fig. 3.150).

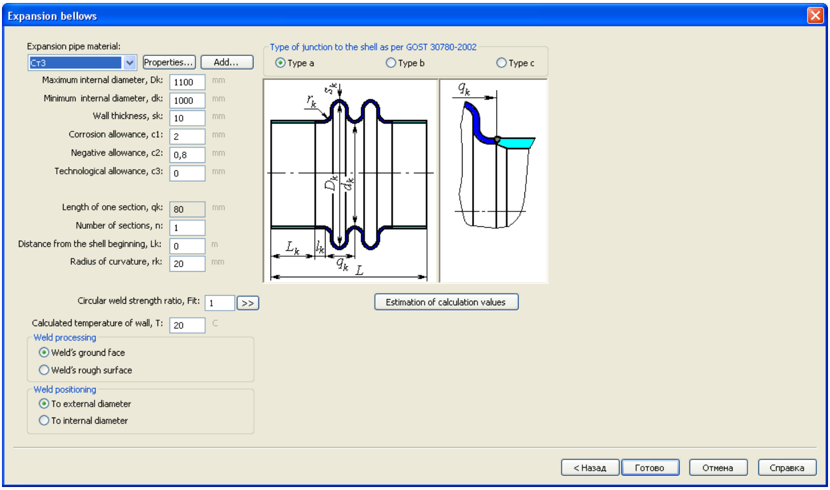


Fig. 3.150 Expansion bellows

At calculation of heat exchanger as per GOST 34233-2017, bellows type with casing is determined according to GOST 30780-2002 (Fig. 3.151).

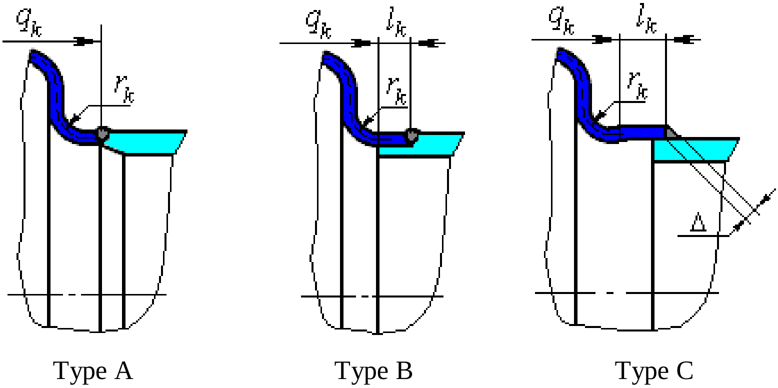
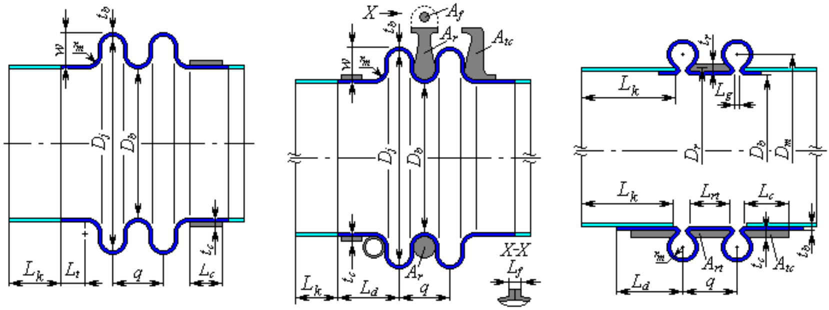


Fig. 3.151 Joint of expansion bellows and casing as per GOST 30780-2002

At calculation of heat exchanger as per ASME VIII-1, possible design variants of expansion joints as defined according to section MANDATORY APPENDIX 26 (Fig. 3.152).



Without reinforcement With reinforcement Toroidal

Fig. 3.152 Design variants of expansion bellows as per ASME VIII-1

3.17.39. Heat Exchanger with expansion box in the casing

To include the expansion box in heat exchanger model, select an appropriate checkmark in the "Expansion box" tab (Fig. 3.153). In the presence of expansion box on the expander, its parameters are set similarly i. 3.17.38.

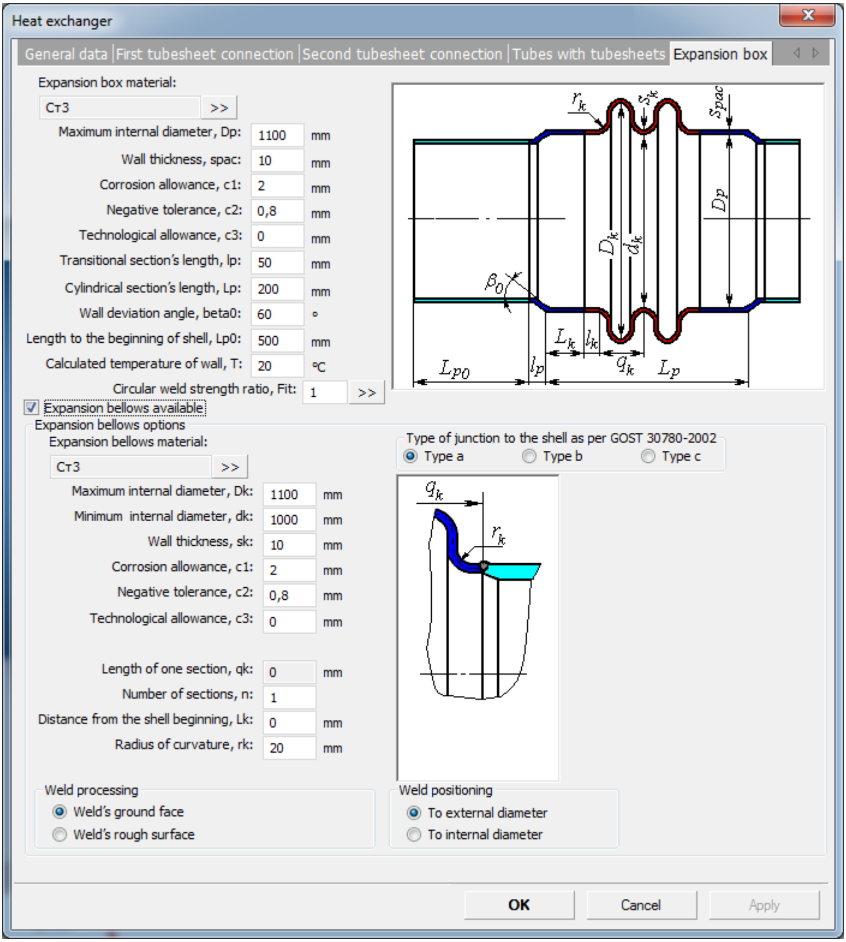


Fig. 3.153 Expansion box

When calculating the heat exchanger, the flexibility of the expansion box is taken into account. If the expansion box is made with bellowed sides, use this option (Fig. 3.154).

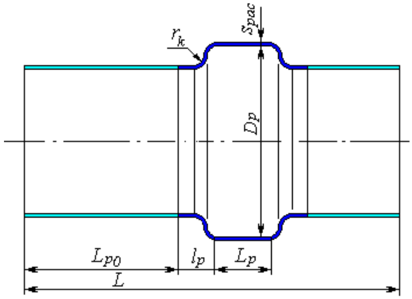


Fig. 3.154 Expansion box with bellows on the sides

Do not use the "Expansion box" option to define the distribution manifold (without cutting the casing section under it) - for this, use the "Cylindrical jacket"component.

3.17.40. Heat Exchanger with U-shaped tubes

Data input is similar to heat exchangers with stationary tube plates. The tube plate must always have a pass partition and tubes must be arranged symmetrically.

Tube sheet for this heat exchanger can be performed similar to i. 3.17.35.

Besides, as per GOST 34233.7-2017, a variant of tube sheet clamped between the flanges is additionally available (Fig. 3.155)

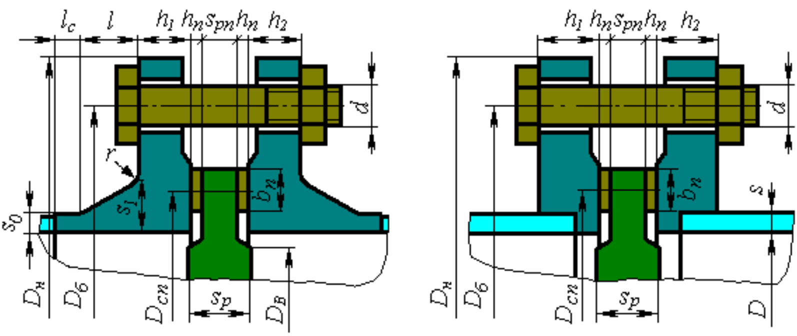


Fig. 3.155 Tube sheet between flanges

In accordance with ASME VIII-1, additional configurations are available (see Fig. 3.156).

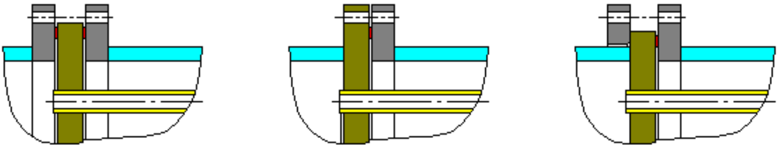


Fig. 3.156 Tubesheet types according to ASME VIII-1

The “Electric immersion heater” option allows the calculation of the heater base plate as a perforated flat cover.

3.17.41. Heat Exchanger with Floating Head

Data input is similar to heat exchangers with stationary tube plates. Properties of the floating head are input instead of second tube plate (Fig. 3.157).

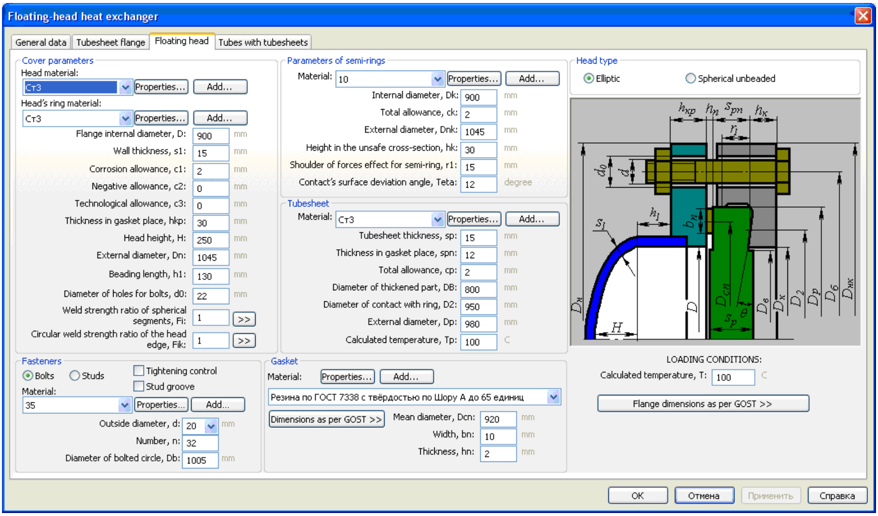


Fig. 3.157 Floating head

At calculation of heat exchanger as per GOST 34233.7-2017, the floating head may include elliptic head and spherical unbeaded head.

Possible variants of floating heads are shown on the Fig. 3.158

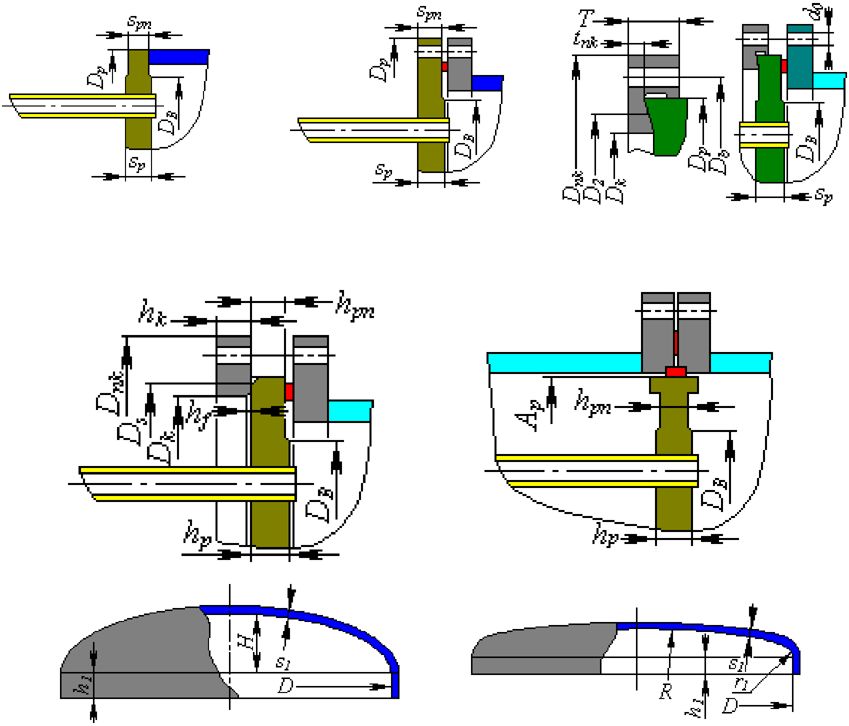


Fig. 3.158 Types of floating heads

Version of head corresponds to section “Bolted heads”.

3.17.42. Air cooled exchanger

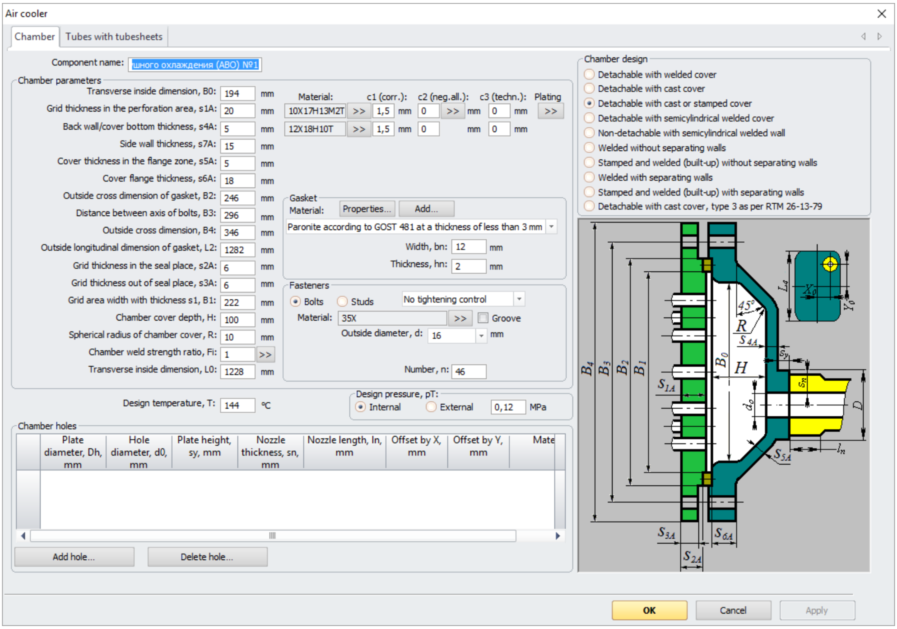
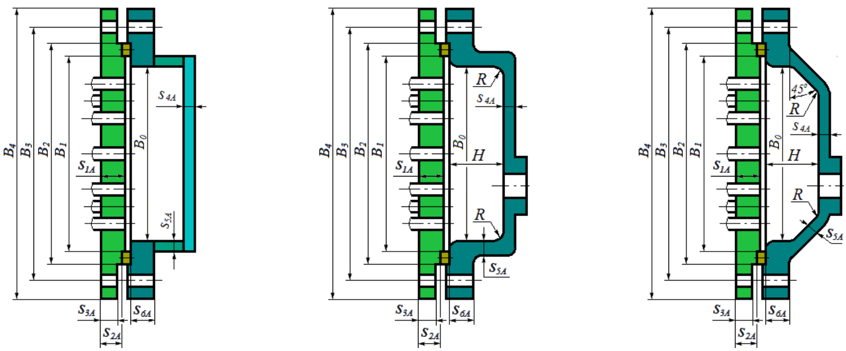
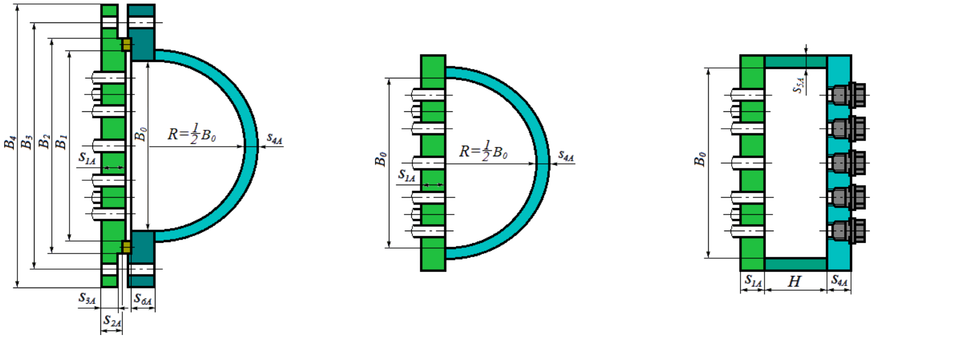


Fig. 3.159. Distribution chamber

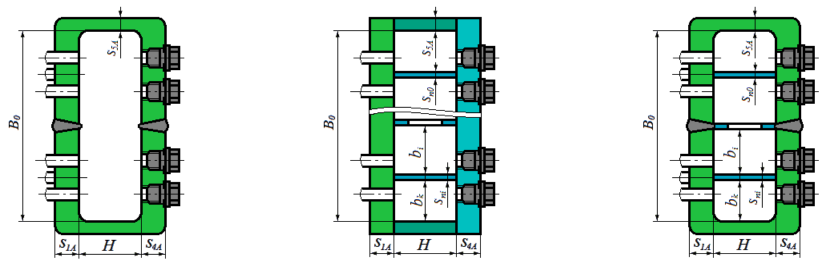
Air-cooled heat exchanger is created as a model component. This component cannot be joined to anything. No other components can be joined to it as well in the current program version. Air cooled exchanger consists of two identical distribution chambers (Fig. 3.160) and tube bundle (Fig. 3.162). Two heat exchanger chambers can be set independently and have a different type. Special nozzle type can be attached to the chamber.



Bolted with welded head	Bolted with cast head	Bolted with cast or stamped head



Bolted with semicylindrical welded head	Non-detachable with semicylindrical welded head	Welded without separating walls



Stamped and welded without separating walls	Welded with separating walls	Stamped and welded with separating walls

Fig. 3.160. Types of distribution chambers

For welded components of heat exchanger, you can assign a negative tolerance, cladding.

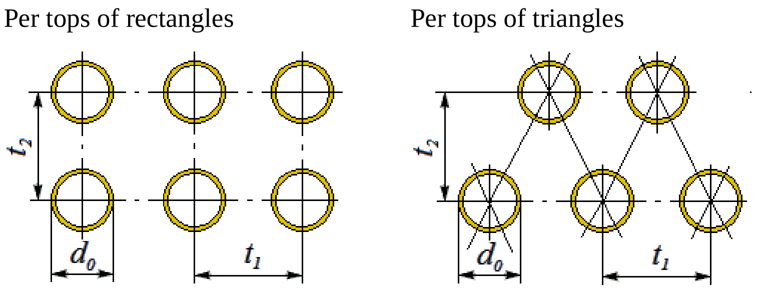


Fig. 3.161. Types of tube bundles

Upon pressing  button, tube bundle parameters can be defined, similar to i. 3.17.35.

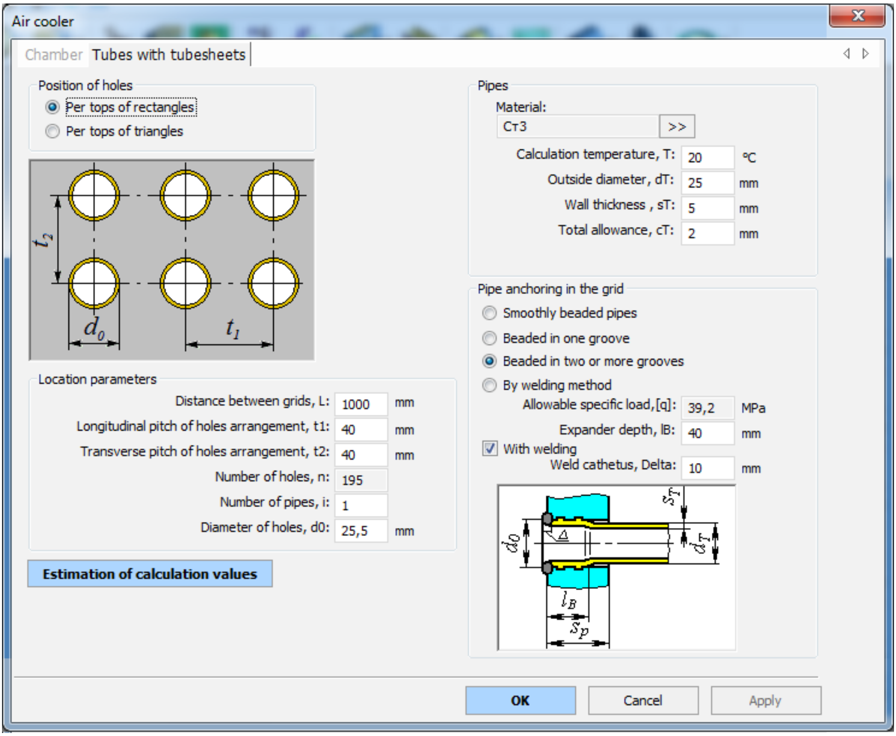


Fig. 3.162. Tube bundle

If there are screw plugs in the chambers (Fig. 3.163) they can be calculated according to [34].

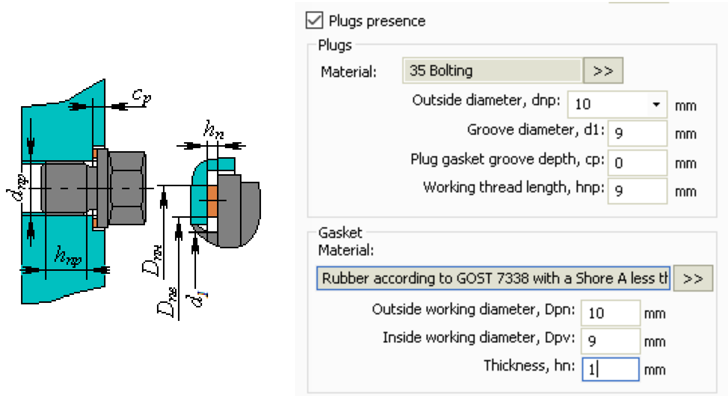


Fig. 3.163. Screw plugs in the chamber

The outer frame can be set according to Fig. 3.164 (it is taken into account only in the visualization of the model and in the calculation of the metal consumption table).

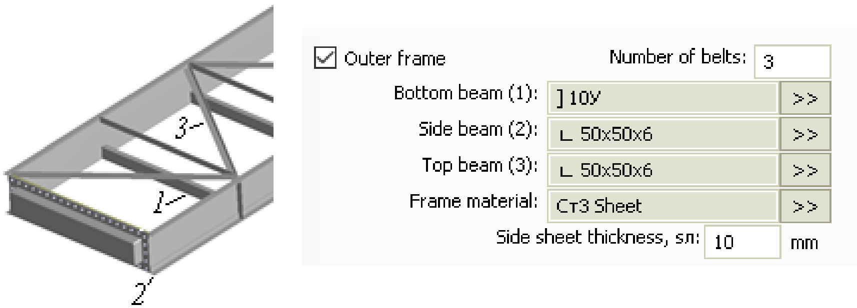


Fig. 3.164. External air cooler frame

3.17.43. Nozzle of the air cooler chamber

This component can be attached to the back wall of the cast/forged air cooler chamber, to the rear surface of the cylindrical chamber, to the back or side surface of the box chamber. Input data of tie-in into the cylindrical chamber are set similarly to the component "Nozzle". For tie-in into the flat wall, the dialogue looks like Fig. 3.165

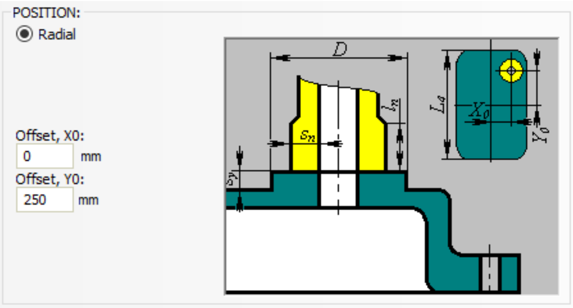


Fig. 3.165 Nozzle of the air cooler chamber

3.17.44. Cylindrical jacket

A cylindrical jacket can be joined to any cylindrical shell of the existing model (Fig. 3.166). Component name, code of standards, material, dimensions, weld strength factors and load properties for jackets are set in the same way as those for cylindrical shells. Jacket placement within the model is determined by the jacket's adjoining component and the distance from left (bottom) edge (toward Z-axis). Supports, nozzles, stiffening rings and other components can be adjoined to the jacket. Jacket pressure, p_2, is transferred to adjoining components, and vice versa. The jacket cannot be placed outside the parameters of the shell on which it is placed.

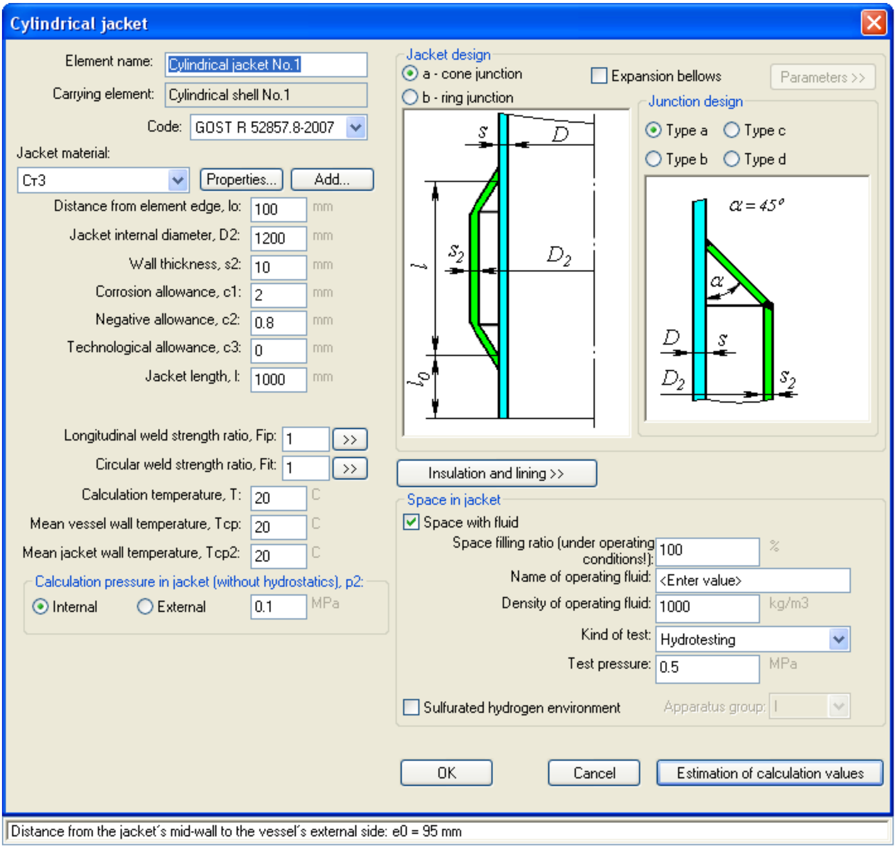


Fig. 3.166 Cylindrical jacket

Properties of inner jacket environment test properties without supporting shell can be input.

Structure of jacket and shell joint is determined according to GOST 34233.8-2017 (GOST 25867-83) (see Fig. 3.167).

Volume and weight calculation of the jackets content is possible only using the fill factor.

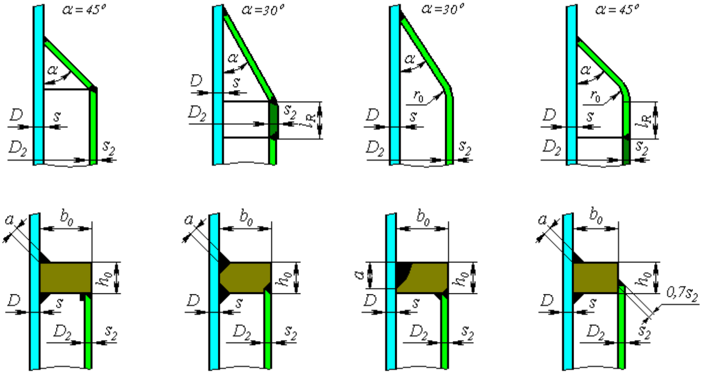


Fig. 3.167 Jacket types as per GOST

Cylinder jackets can have bellows (Expansion bellows) to reduce loads from temperature deformations.

It is possible to calculate jacket as per ASME VIII-1. In this case, variants of connection with casing specified in Fig. 3.168 are taken.

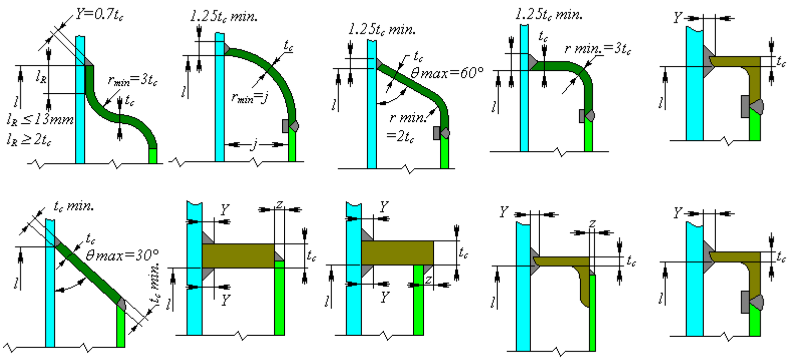


Fig. 3.168 Jacket types as per ASME VIII-1

3.17.45. U-shaped jacket

U-shaped jackets are created using a multi-page dialog and include the following components:

- Vessel shell;

- Jacket shell;
- Vessel head;
- Jacket head.

Jacket and vessel joint data are input similarly to those for cylinder jackets. Jacket and vessel head properties are input in the same way as those for dished heads.

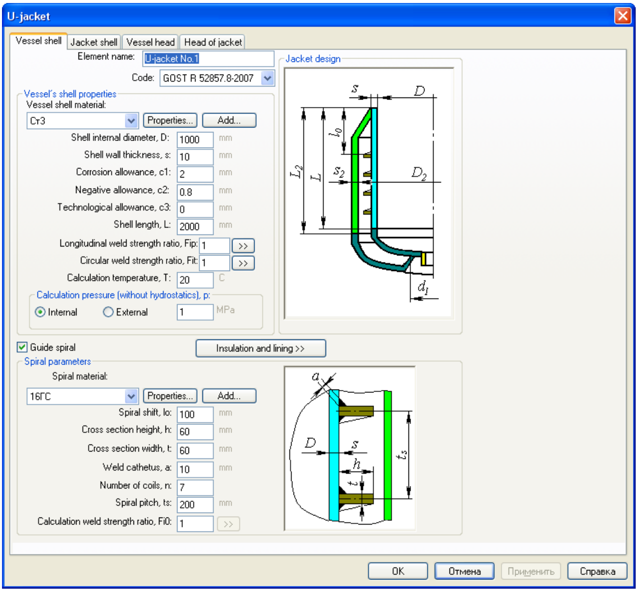


Fig. 3.169 U-shaped jacket: vessel shell

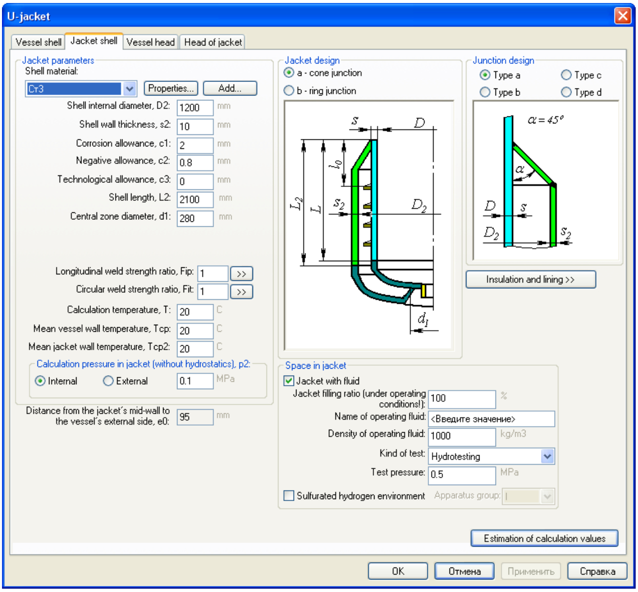


Fig. 3.170 U-shaped jacket: jacket shell

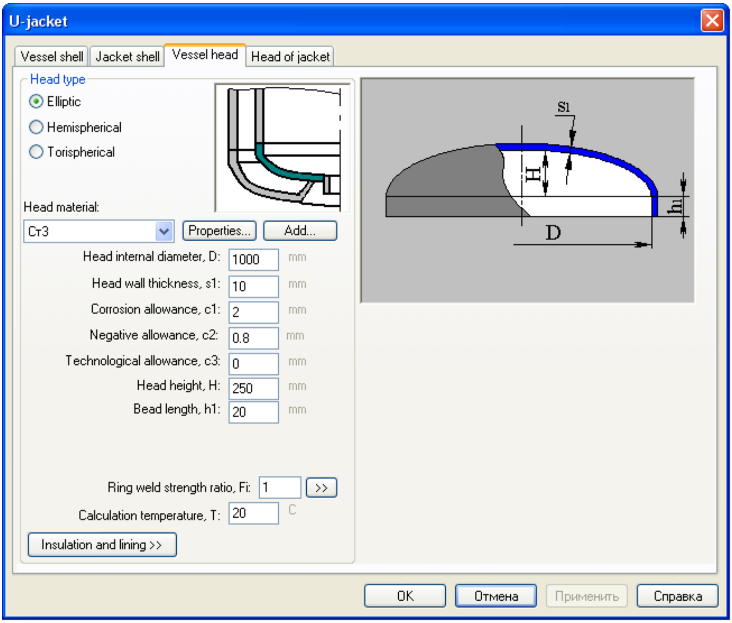


Fig. 3.171 U-shaped jacket: vessel head

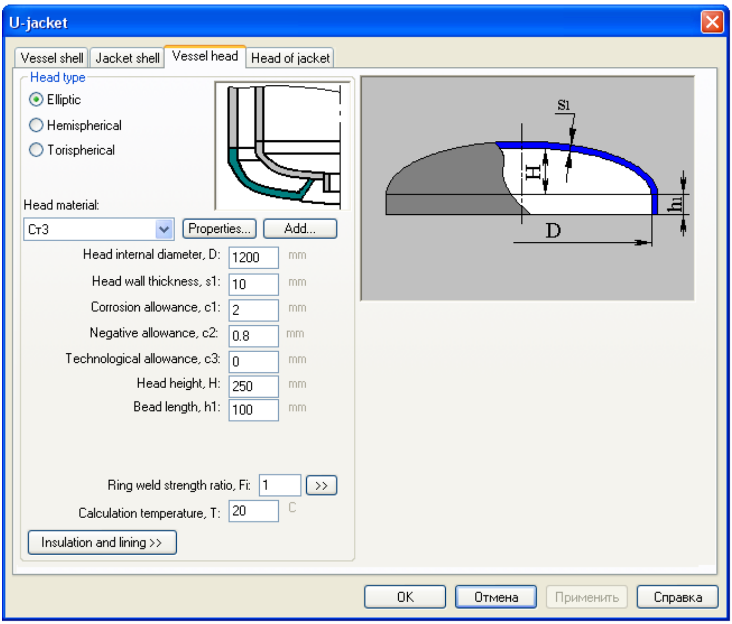


Fig. 3.172 U-shaped jacket: jacket head

3.17.46. Partial jacketing

Partial jacketing data are input similarly to those for cylindrical jackets.

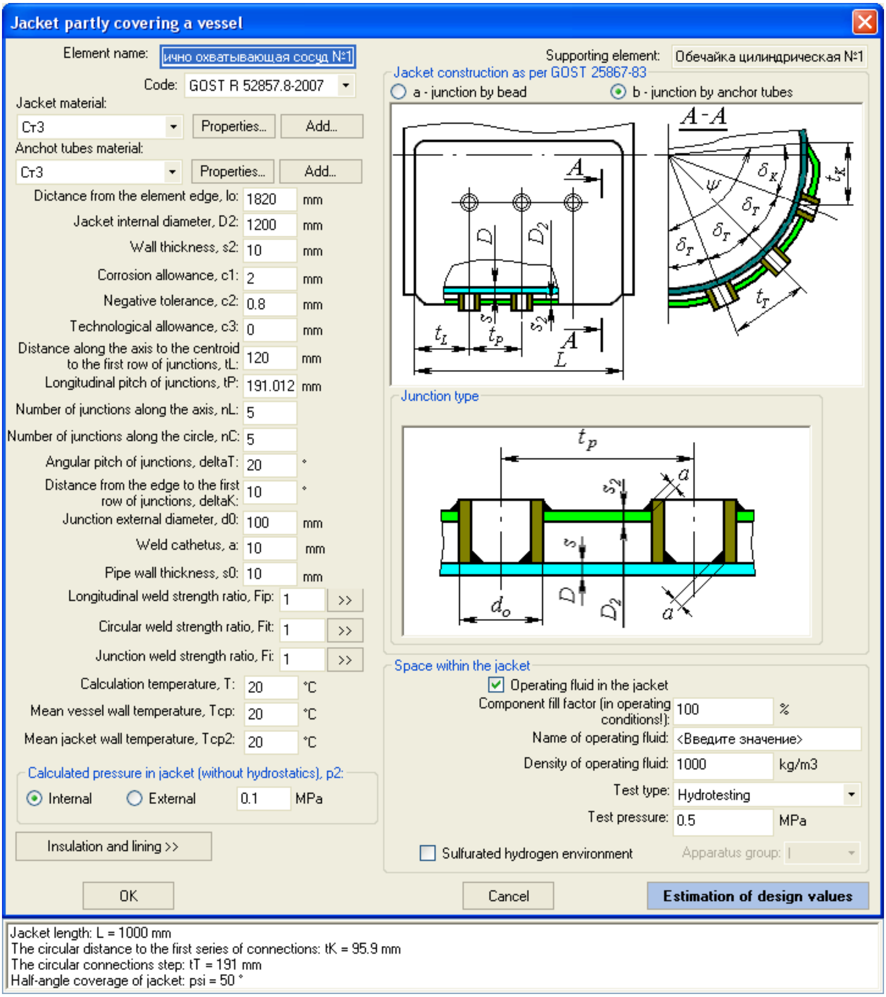


Fig. 3.173 Partial jacketing

Jacket and shell joint type is determined according to GOST 34233.8-2017 (GOST 25867-83) (see Fig. 3.174).

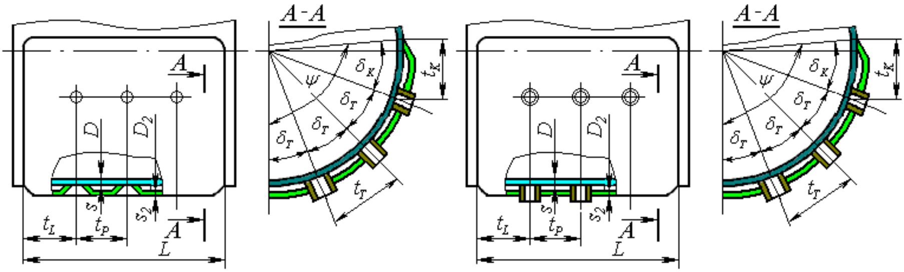


Fig. 3.174 Jacket and vessel body joint types

3.17.47. Half-pipe coil jacket

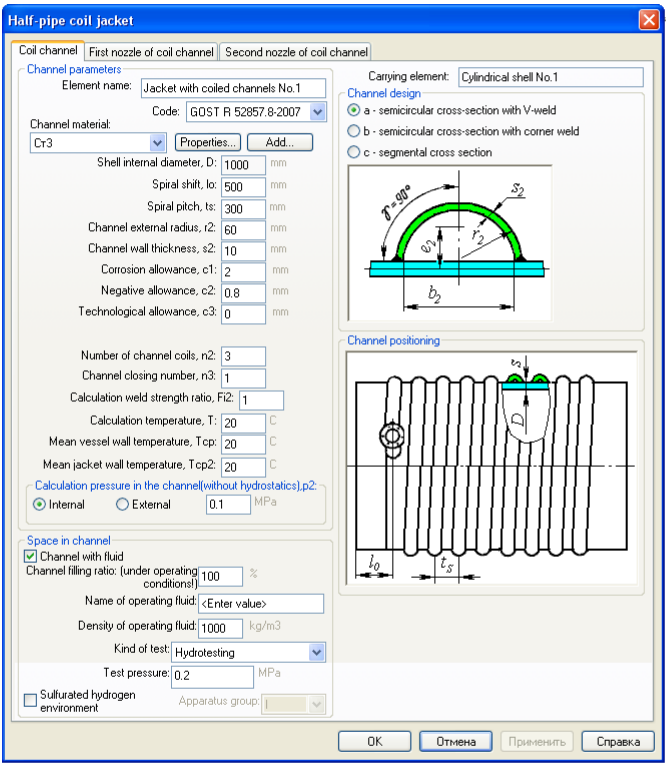


Fig. 3.175 Half-pipe coil jacket

Spiral jacket data are input similarly to those for cylindrical jackets. During analysis, the coils can be treated as reinforcement of supporting shell by a system of stiffening rings. Coil types are determined by GOST 34233.8-2017 (GOST 25867-83) (see Fig. 3.176). Nozzles are automatically placed at coil ends.

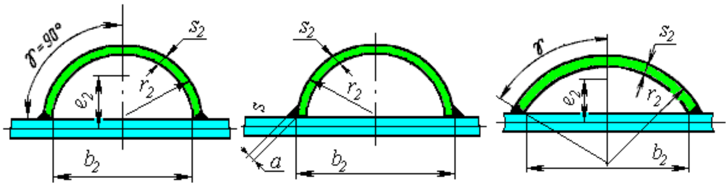


Fig. 3.176 Coil types

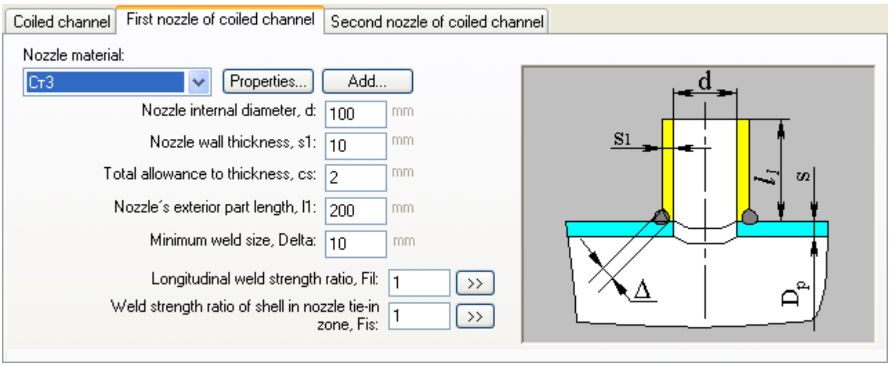


Fig. 3.177 Coil nozzle

3.17.48. Half-pipe battery jacket

Half-pipe coil jacket data are input similarly to those for spiral jacket. In accordance with GOST 34233.8-2017 (GOST 25867-83), this jacket is not considered as a system of stiffening rings.

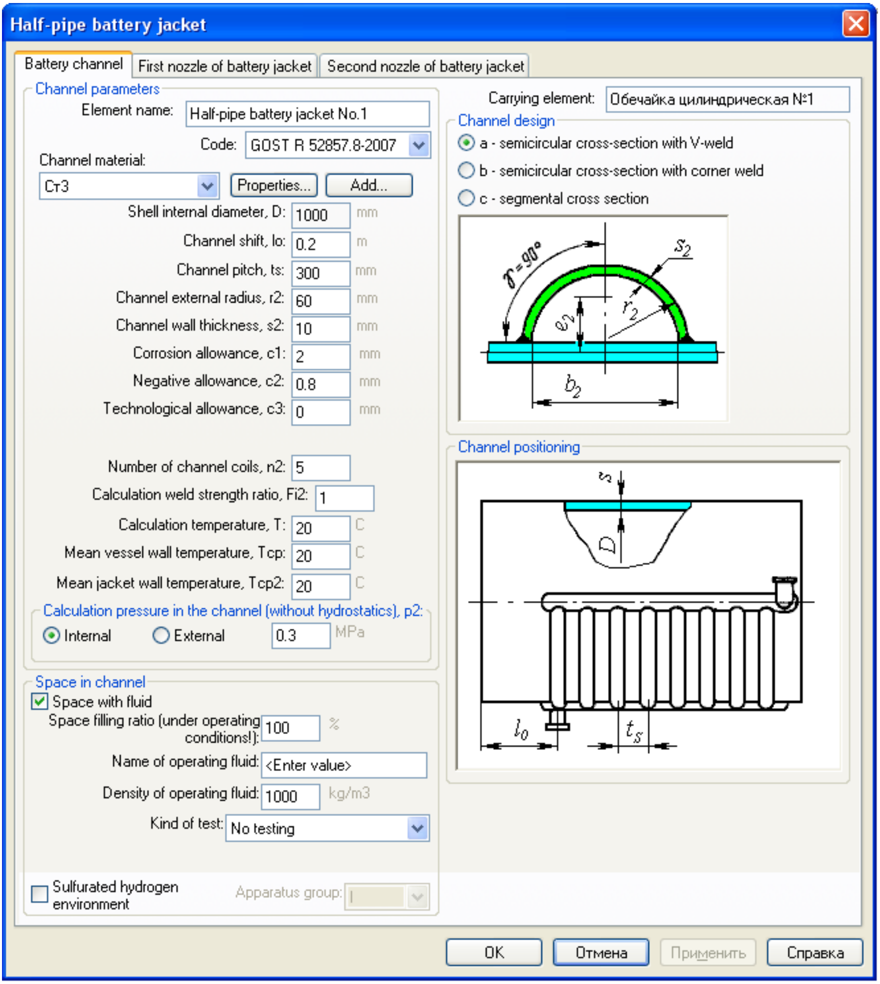


Fig. 3.178 Half-pipe battery jacket

3.17.49. Jacket with longitudinal channels

Jacket with longitudinal channels can be connected to cylindrical shell or conical transition.

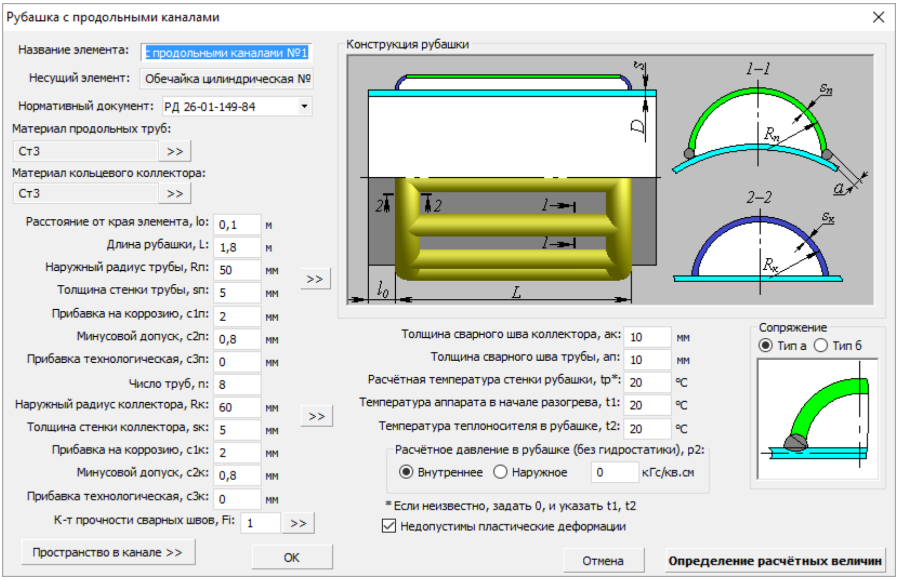


Fig. 3.179 Jacket with longitudinal channels

Parameters of channel cavity are specified by pressing “Space in channel" button, similar to 3.17.1.13.

Types of conjunction are set as per [51] and indicated in Fig. 3.180.

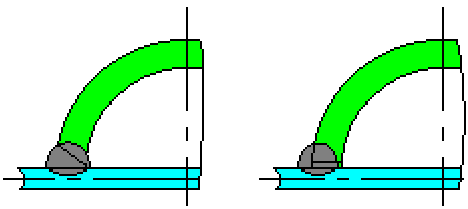


Fig. 3.180 Types of conjunctions

Parameters of pipes and collector (radius, thickness, negative allowance) can be selected from the grade using button >>.

If the wall calculation temperature tp, is unknown, it is necessary to specify“0” instead of it and set temperatures t_1, t_2.

Option **“Inadmissible plastic deformations”** is used in the presence of brittle coatings, possibility of corrosion cracking, etc.

3.17.50. Convex bulk

This component can be used in horizontal, vertical and column vessels for separation of volumes with different pressures and fillings. In the course of model building, it can be joined to the other components and inserted between them like a cylindrical shell, but during calculation it should be always placed between the other components. A separating wall creates a new volume, calculation of filling for which is performed separately. Filling parameters and propertied of the fluid inside this volume are also set in the dialog of separating wall. Daughter components, pressure in which is transferred depending on the separating wall orientation, can be joined to the separating wall.

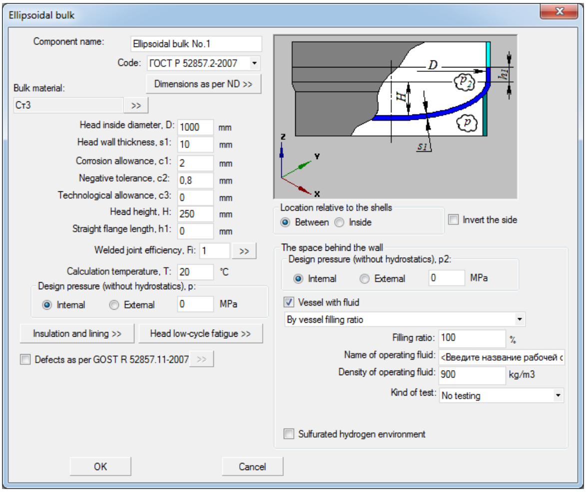


Fig. 3.181 Ellipsoidal bulk

The head of the bulk can be elliptical, spherical or torispherical.

3.17.51. Virtual bulk

This component can be used in the same manner as convex bulk, but without strength and stability calculation of the separating wall itself (for instance, for modeling vessels with layer by layer filling with a heterogeneous medium, coke collectors, etc.).

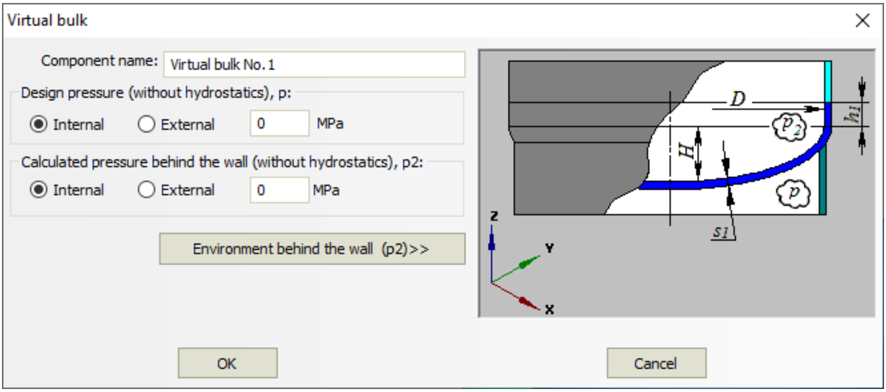


Fig. 3.182 Virtual bulk

3.17.52. Ellipsoidal transition

This component can be used in places of different diameters.

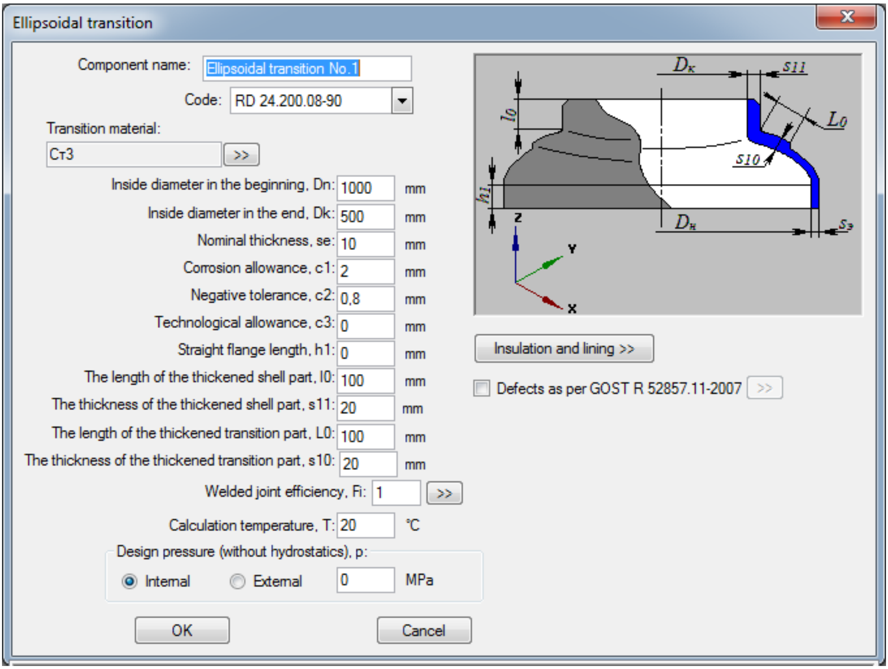


Fig. 3.183 Ellipsoidal transition

3.17.53. Expansion bellows

This component behaves in the model similarly to elbow. Its calculation includes an assessment of strength and stability against pressure and displacements, including the calculation of low-cycle fatigue.

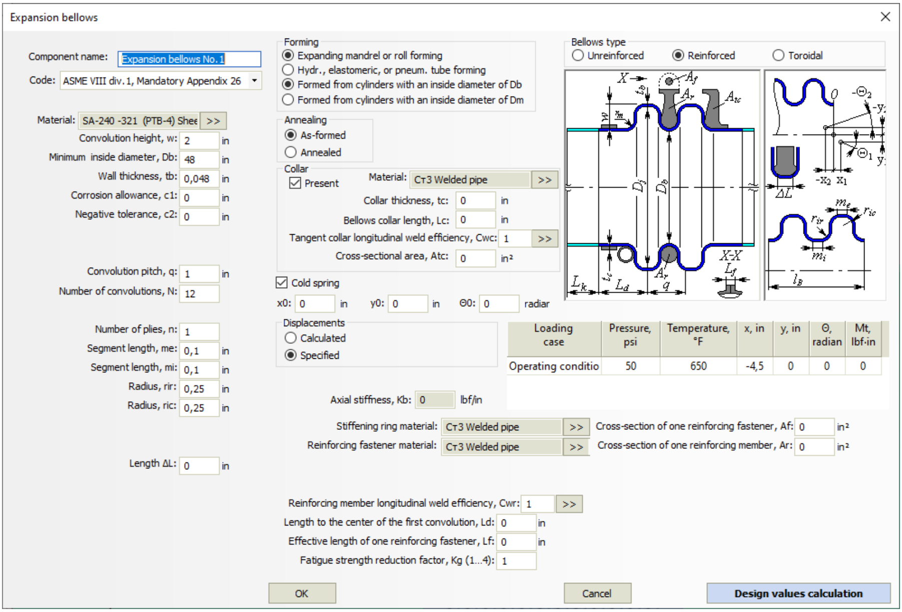


Fig. 3.184 Expansion bellows

Expansion bellows deformations can be calculated automatically, based on the fixing and loading conditions of the model. To do this, use the option “Displacements” – “Calculated”.

The “Cold spring” option allows to specify a prestressed bellows.

For bellows operating as part of a shell (shell-and-tube heat exchanger), the “No calculation” option is available - it allows to specify a non-standard bellows with a known stiffness and take it into account in the calculation of the shell.

3.17.54. Structure

This component can be attached as a child component to the casing shell; its input data are set similarly to column support structure. Its calculation includes stiffness assessment and implementation of steel structure in the beam model as a super-element. The strength of the structure parts (beams) is not currently estimated.

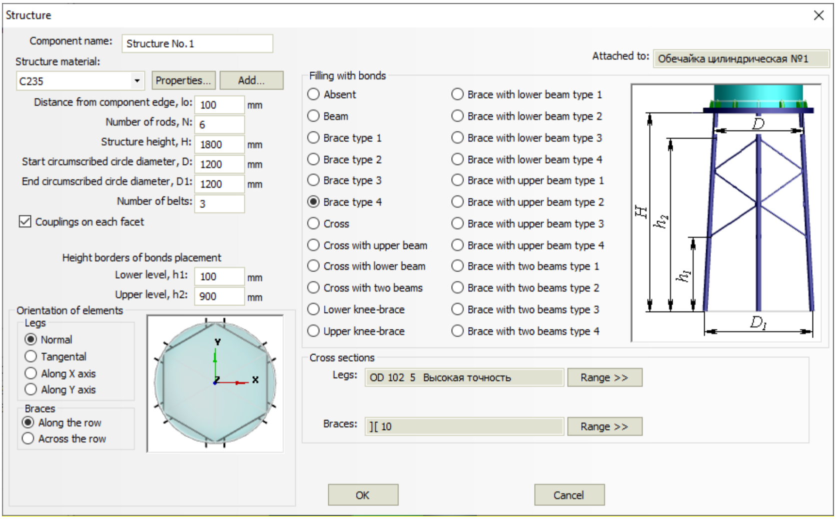


Fig. 3.185 Structure

The steel structure shall be placed entirely within the boundaries of the parent shell. It can also be placed as part of an assembly. An assembly can contain a sequence of structures, in which case their beam models are linked automatically. End points of structures can have links and fastenings.

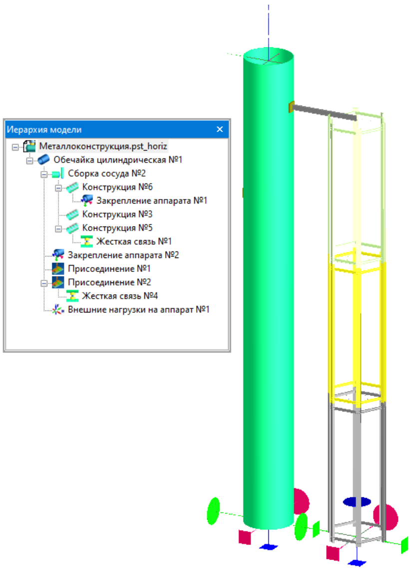


Fig. 3.186. Modeling a stand by structure

3.17.55. Vertical steel tank for oil and oil products

When selecting “Vertical tanks” (Fig. 3.6), an component “Tank” is created automatically in the model, and the dialog with its data opens.

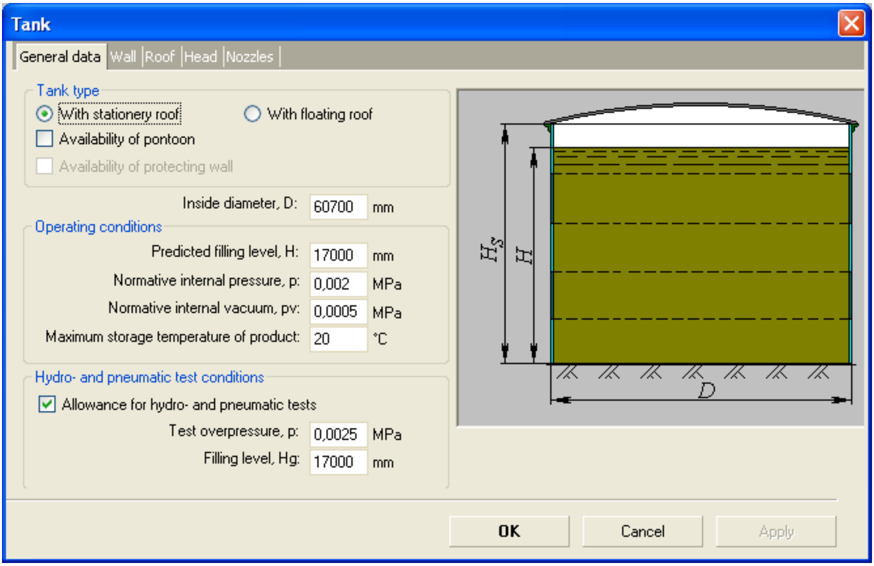


Fig. 3.187 Tank general data

Currently a calculation of the tanks with stationary and floating roof is implemented. Calculation of tanks is available according to the following codes:

- STO-SA-03-002-2009 [54]
- GOST 31385-2016 [31]
- API 650-2020 [75]

Normative internal pressure and internal vacuum are set above the surface of the product (without without regard to hydrostatic pressure). When a checkbox “Pontoon presence” is selected, a weight of the pontoon being at the filling height, is considered (calculation of the pontoon itself is not provided yet).

Component “Tank” cannot be deleted from the model, but can be edited; besides, some daughter components can be connected to it. As the daughter components, the following can be taken:

- Stiffening rings (are connected to the wall);
- Nozzles;

- Service platforms (are connected to the wall);

Lumped masses (are connected to the wall and stationary roof, are divided into metal structures and equipment, are considered in different ways at calculation.

Tank wall parameters can be set by selecting Next > button.

3.17.55.1 Tank wall

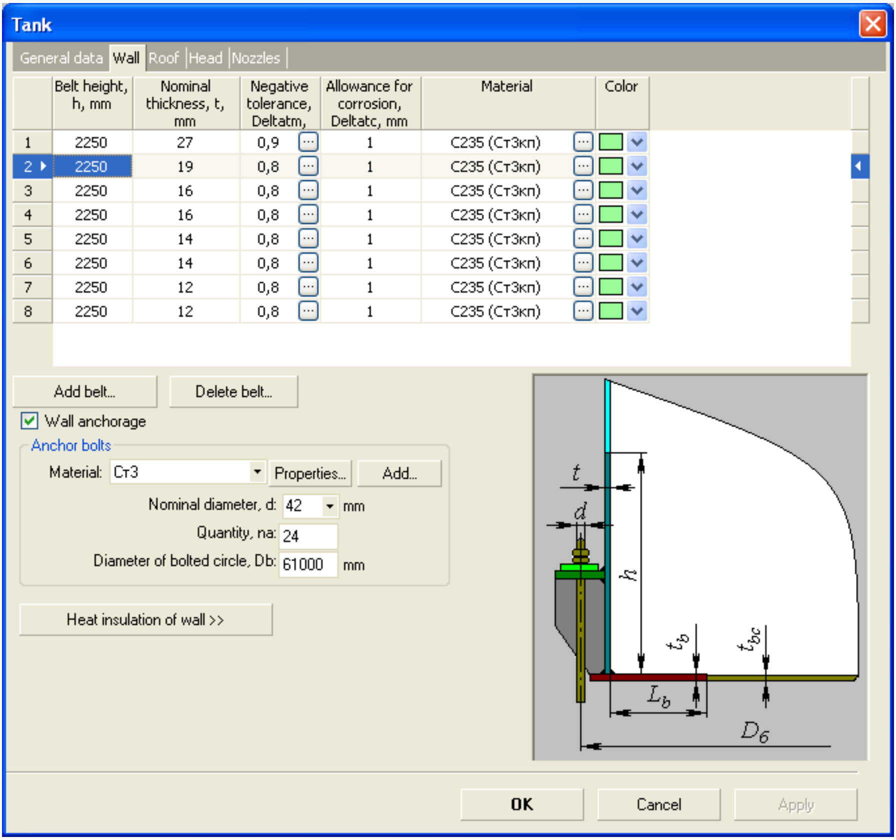


Fig. 3.188 Tank wall

Height, thickness, negative allowance, corrosion allowance and material are set for each belt. Negative allowance and material can be selected by … buttons, in accordance with [53].

If additional anchoring of the wall to the foundation is required, anchor bolts can be specified.

Thermal insulation parameters are set in accordance with clause 3.17.1.10.

When modeling a tank with a floating roof, it is necessary to specify the data of the upper wind ring (in addition to it, you can specify an arbitrary number of intermediate rings according to clause 3.17.18).

When calculating API-650, you can select the calculation method according to the code (“1 Foot Calculation Method” is simplified, “Variable Design Point” is more accurate).

Tank roof parameters can be set by selecting 

button.

3.17.55.2 Tank roof

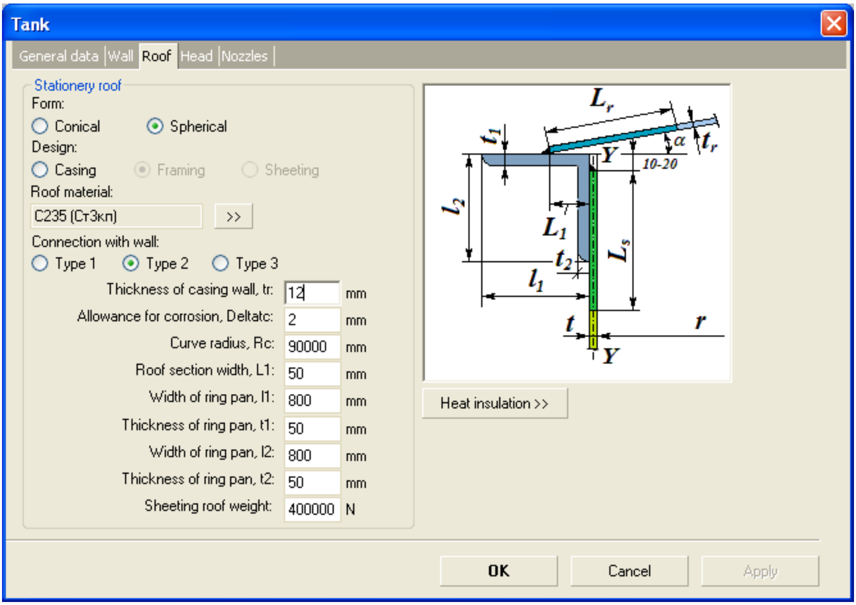


Fig. 3.189 Tank roof

Style, material and type of connection with wall are set.

When calculating according to API 650, the following options are available for joining the roof to the wall (Fig. 3.190):

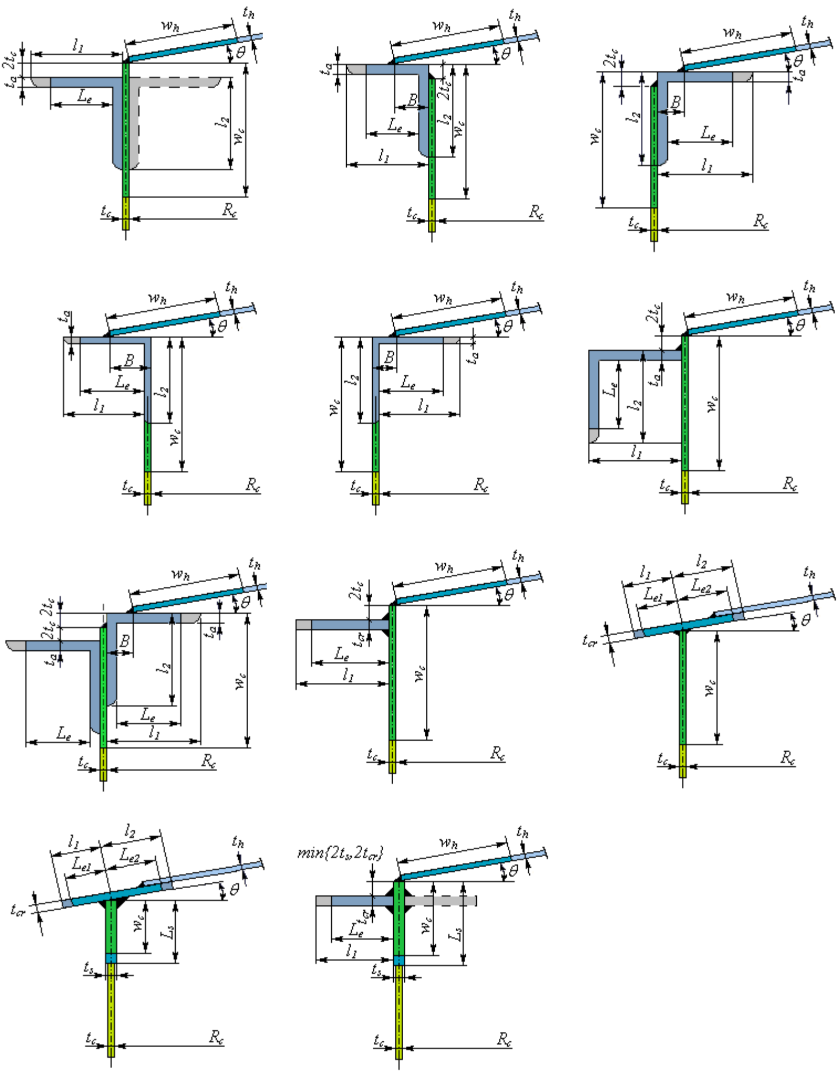


Fig. 3.190. Roof-to-wall joining types as per API 650

The calculation of the supported roof strength and buckling is **not performed** in the current version of the program; only weight loads due to metal structures are taken into account. The weight of the frame roof can be specified in various ways:

- manually (weight value is specified);

- by the attached frame (it is necessary to prepare the frame model in a third-party CAD system similarly to the "Custom equipment" component, Fig. 3.191).
- by supported roof designer tool.

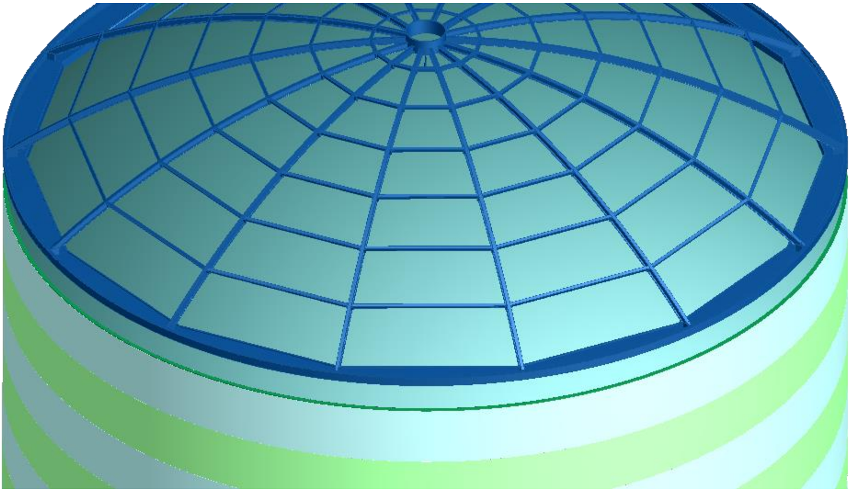


Fig. 3.191 Imported roof frame model

Tank bottom parameters can be set by selecting Next > button.

3.17.55.3 Supported roof designer

This tool allows to create a mesh of beam profiles. The created mesh is "stretched" onto the generatrix of the roof, the weight loads due to all beam elements of the frame are summed up. In addition, the tool allows to create a complex beam-shell finite element model of the tank and export it to an APDL file for strength and stability studies in the Ansys program.

Objects that the designer operates on:

- Nodes (nodal points) - indicate the places where the frame elements are joined together and with the roof shell. The nodal points split the roof shell into 4- nodal finite elements;
- Connecting nodes (nodes with the attribute "Roof-wall connection"). This nodal point is involved in the modeling of the wall (from the circular sequence of such nodes, shell 4-nodal elements of the wall are "grown" down to the foundation). It is desirable to arrange the connecting nodes evenly and assign their coordinates as accurately as possible along the circumference of

the wall. An error in the value of the connecting nodes can lead to a significant distortion of the solution;

- Support nodes (nodes with the "Roof-column connection" attribute). In this case, a beam element is "grown" from the nodal point to the foundation, modeling a column of a given section;
- Free nodes - nodes where beam elements do not join. Such nodes are used to control the mesh (the roof shell is divided into shell elements using all specified nodes);
- Element – section of a beam, connecting two arbitrary nodes with given cross-section;
- Section – profile of the element, which can be selected from the database. The sections are named ("Rafters", "Beams", etc.), but this naming is conditional and does not necessarily reflect the functionality of the element. The exception is the "Column" section – support nodes always generate a vertical element of just such a section.

When opening the designer, a circular selection is displayed in the Cartesian axes, corresponding to the diameter of the roof in the plan, as well as the area of the central hole (if the "Skylight" option was activated).

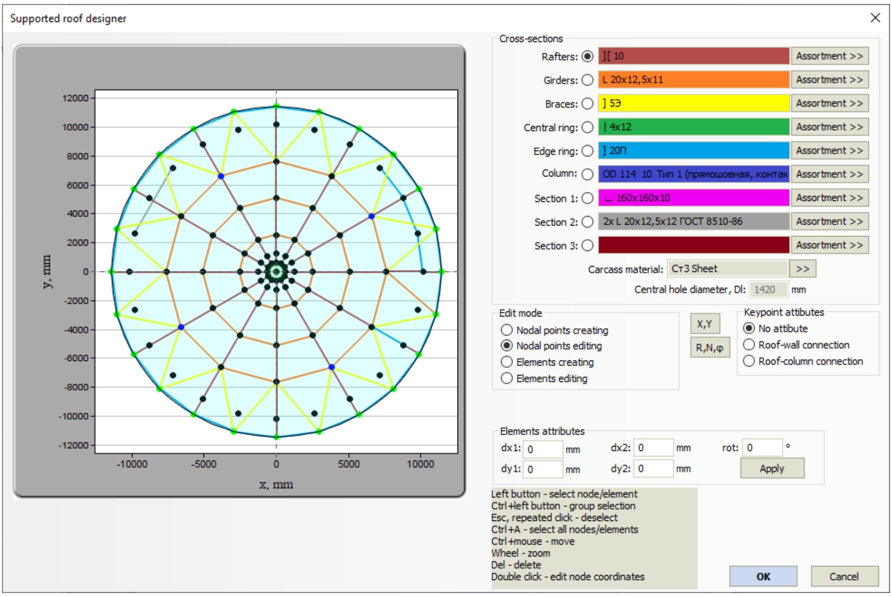


Fig. 3.192. Supported roof designer

The sequence of creating a frame plan:

1) Create nodal points (“Nodal points creating” mode) – for example, by Cartesian coordinates (X, Y), in the form of a circular array (R, N, φ) or by clicking the left mouse button (not recommended, but acceptable, in this case, the coordinates of the node will be taken approximately);
2) Connect nodal points with beam elements (“Elements creating” mode) – when two nodes are sequentially selected, they are connected by an element of the selected section;
3) Assign attributes to the nodes of support and connecting;
4) Assign profile sections.

Nodes can be edited at any time (including after the creation of elements) - in this case, the element grid is not destroyed. For this, the “Nodal points editing” mode is provided.

Node attributes can also be changed at any time (group editing is available by selecting with the Ctrl key).

The section of an element can be changed using the ”Elements editing” mode (group editing is available).

Element attributes are used to fine-tune the parameters of a beam element (*dx, dy* - displacement of the cross section at the end point, *rot* - rotation of the section by a given angle).

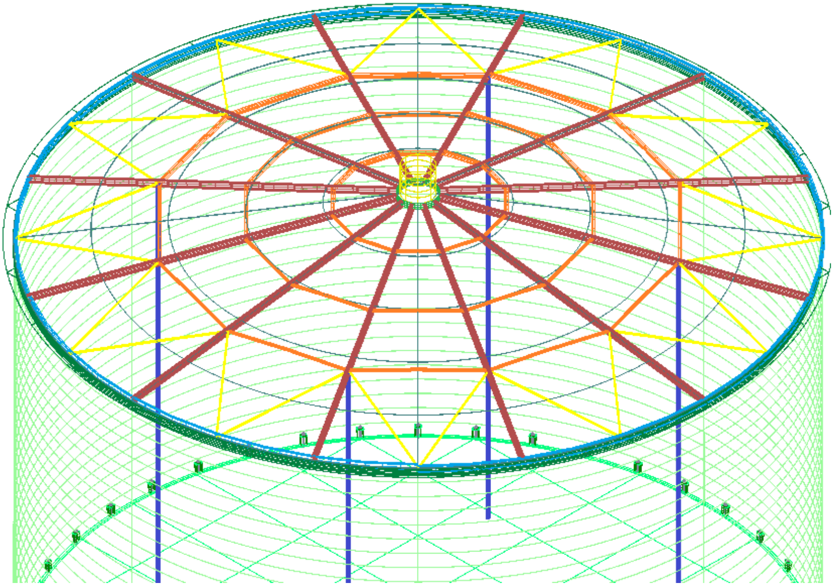


Fig. 3.193. Roof frame model

Based on this information, the program automatically generates a finite element model. The roof shell and wall belts are divided into 4-nodal shell elements, the frame and stiffening elements (rings, columns) are represented by beam elements. Shell elements in the center hole (*Dl*) area are not created. The model can be loaded and exported to an APDL file (see section 3.19.1 for details).

3.17.55.4 Tank bottom

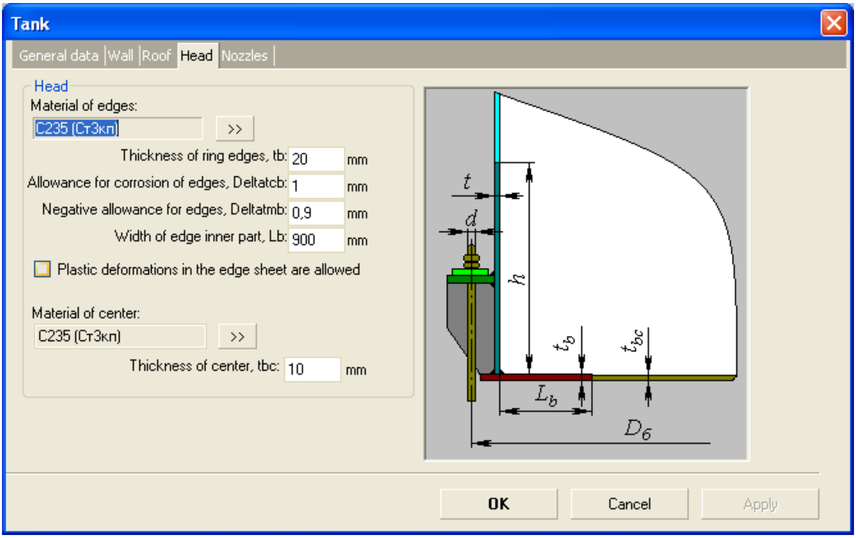


Fig. 3.194 Tank bottom

Information on the nozzles in the wall and stationary roof is entered by selecting Next > button.

3.17.55.5 Tank nozzles

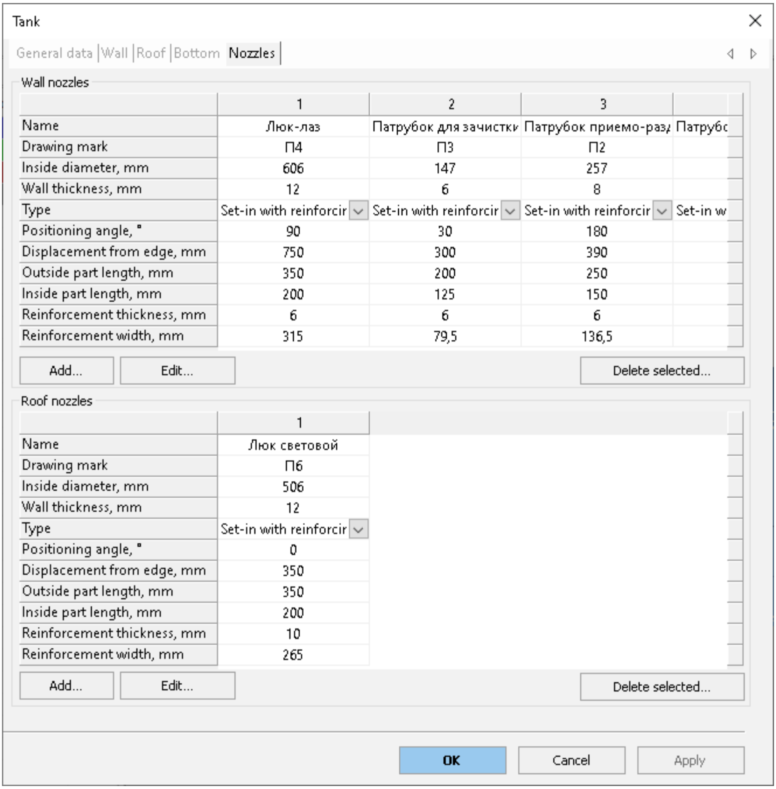


Fig. 3.195 Tank nozzles

For each list of nozzles the following operations are available:

- Adding of nozzle to the wall/roof (“Add”).If some of already created nozzles is highlighted in the list, then data of a newly created nozzle is copied from the selected one; Nozzle data editing window is opened automatically; some data can be corrected right in the list;
- Nozzle editing (command “Edit”);
- Deleting nozzles one by one or group; all selected nozzles are deleted.

Then all added nozzles can be edited and deleted as ordinary components of the model.

3.17.56. High pressure cylinder

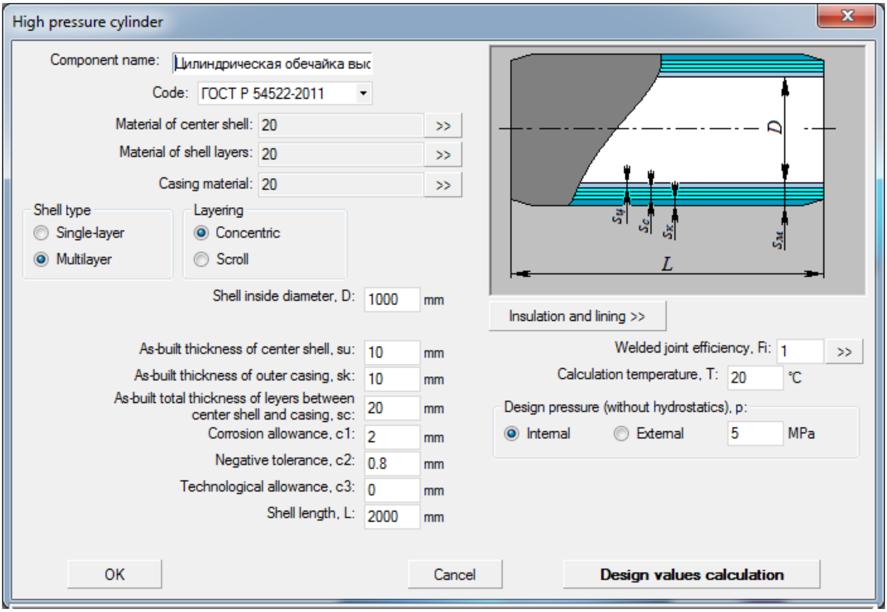


Fig. 3.196 High pressure cylinder

Component name, code, material, geometry and weld strength ratios are assigned equally to cylindrical shell (i. 3.17.2). Both a single-layer and a multilayer shell can be assigned. At that, for multilayer shell you should select a type of layers positioning (concentric or scroll).

3.17.57. Ellipsoidal high pressure head

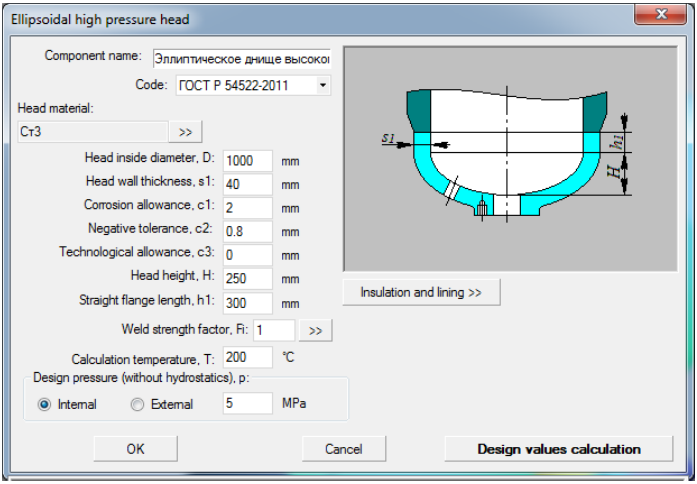


Fig. 3.197 Ellipsoidal high pressure head

3.17.58. High pressure flat head

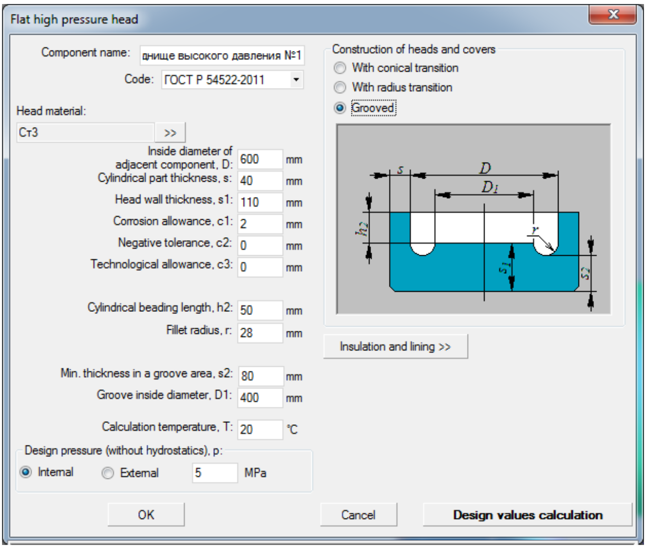


Fig. 3.198 High pressure flat head

It is possible to assign three types of high-pressure flat heads construction: with conical and radial transition, as well as with groove.

3.17.59. Spherical unbeaded high pressure head

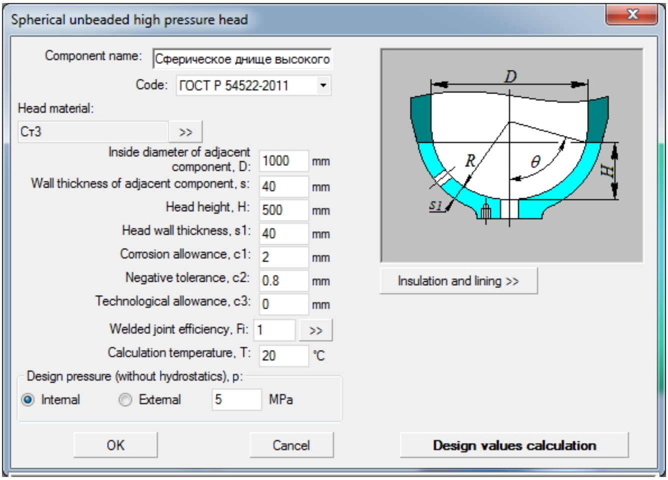


Fig. 3.199 Spherical unbeaded high pressure head

3.17.60. Bolted high pressure flat head

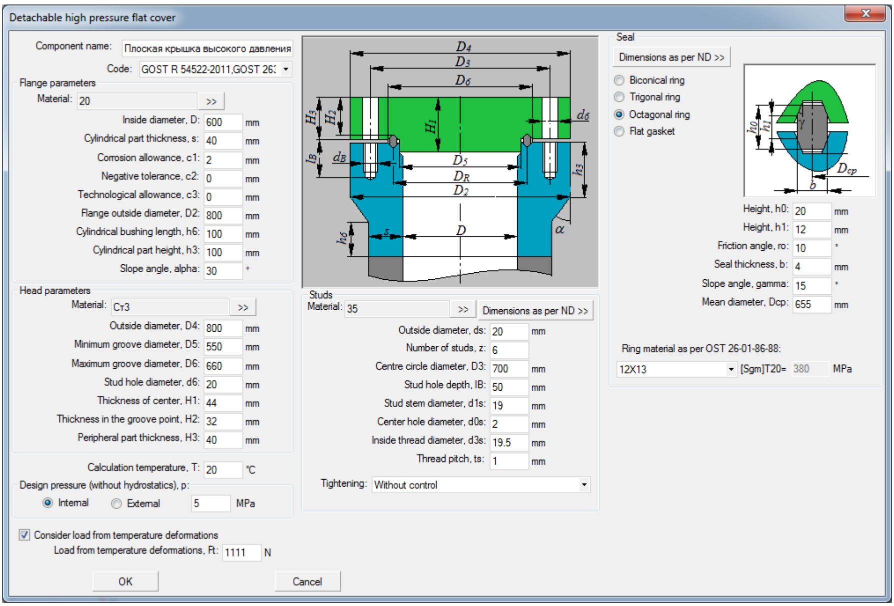


Fig. 3.200 Bolted high pressure flat head

3.17.61. Bolted high pressure spherical head

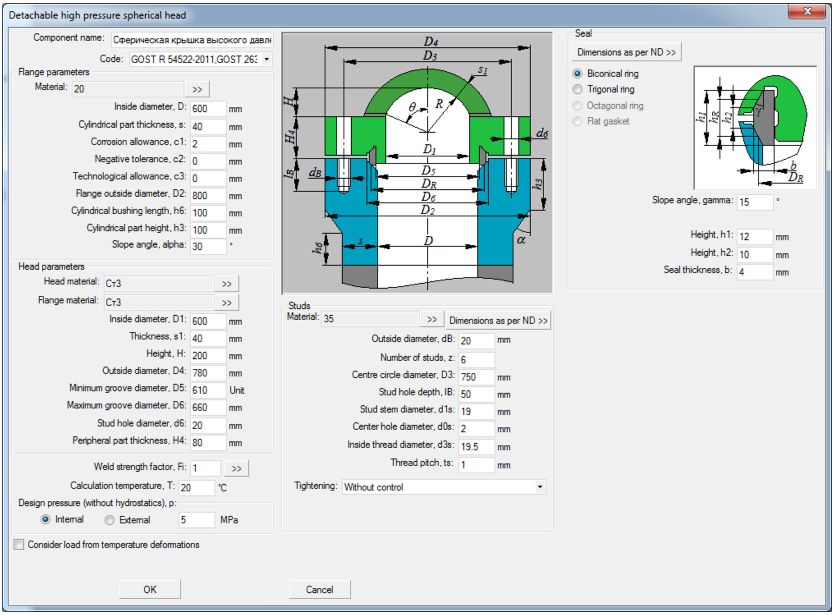


Fig. 3.201 Bolted high pressure spherical head

3.17.62. High pressure nozzle

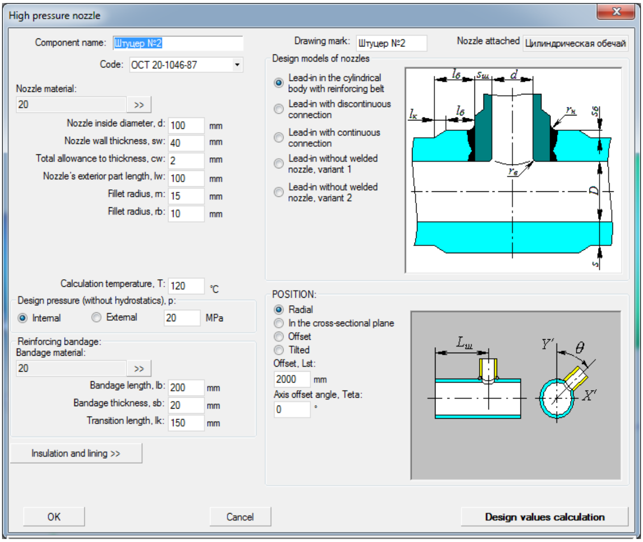


Fig. 3.202 High pressure nozzle

3.17.63. High pressure flange joint

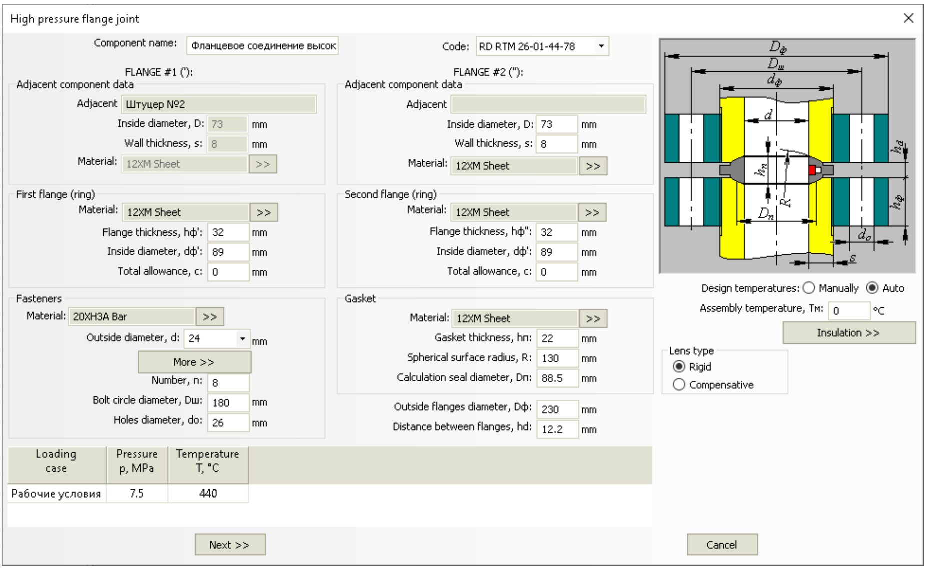


Fig. 3.203 High pressure flange joint

This component can be attached to a high pressure shell or high pressure nozzle.

3.17.64. High pressure bend

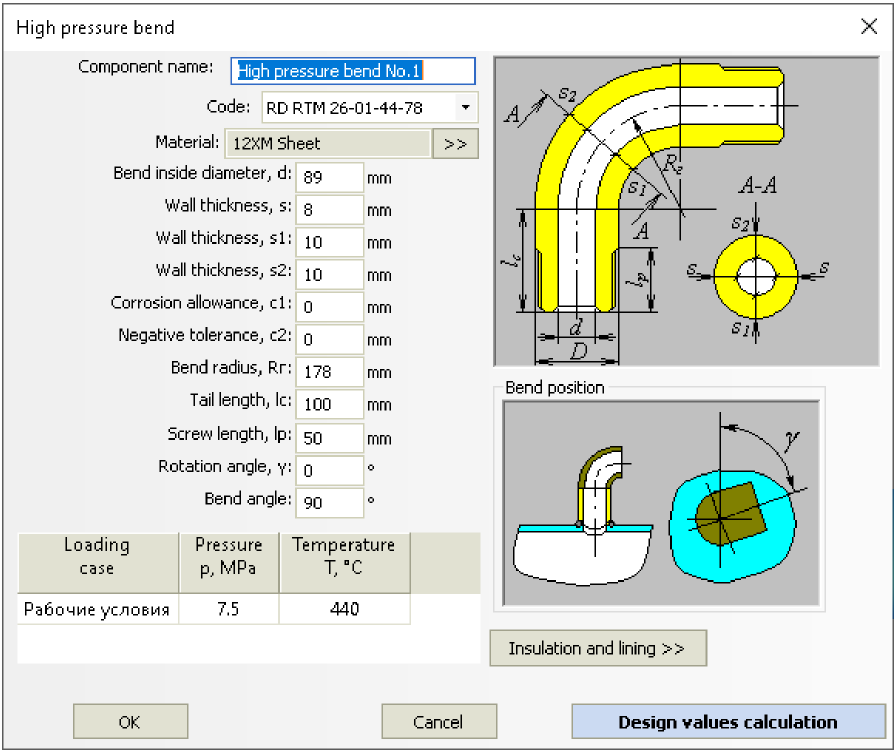


Fig. 3.204 High pressure bend

This component is used to simulate the piping of the high pressure vessels.

3.17.65. Viewing window in the boss

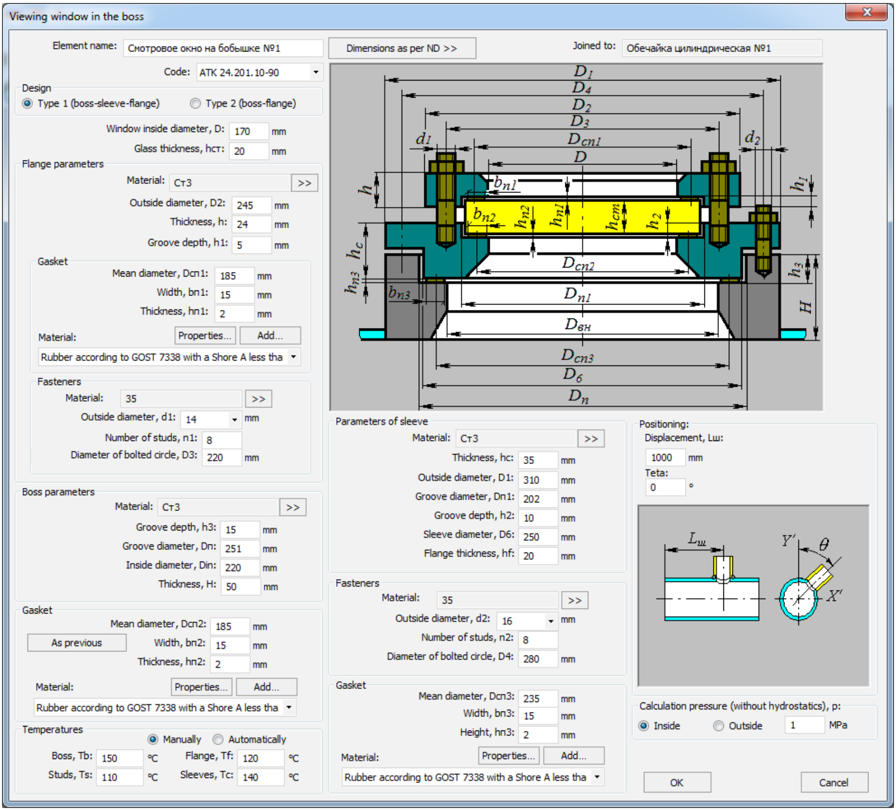
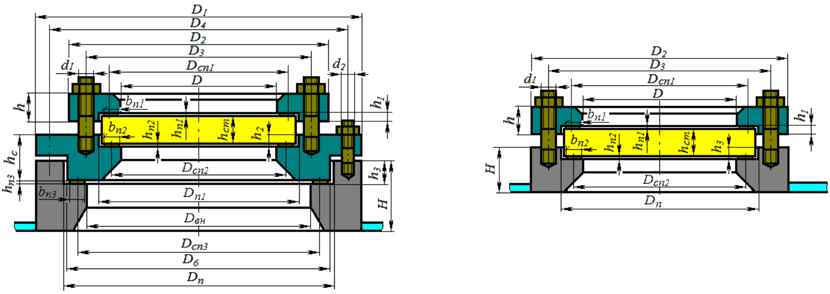


Fig. 3.205 Viewing window in the boss

This component can be joined to cylindrical shell or elliptic head. Possible variants of structure:



Boss-sleeve-flange Boss-flange

Fig. 3.206 Viewing window types

Gasket properties, temperatures and fasteners parameters setting is performed similarly to the flange connection (only soft gaskets can be used).

Window position on the bearing component is defined similarly to the nozzle.

Button Dimensions as per ND >> ("Sizes as per codes") enables selection of standard variants of component from database.

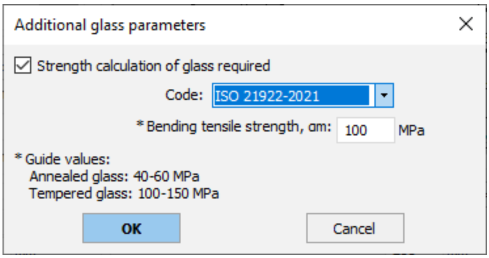


Fig. 3.207 Additional glass parameters

Using the button >> it is possible to set additional glass parameters for strength calculation (Fig. 3.207).

3.17.66. Viewing window in the nozzle

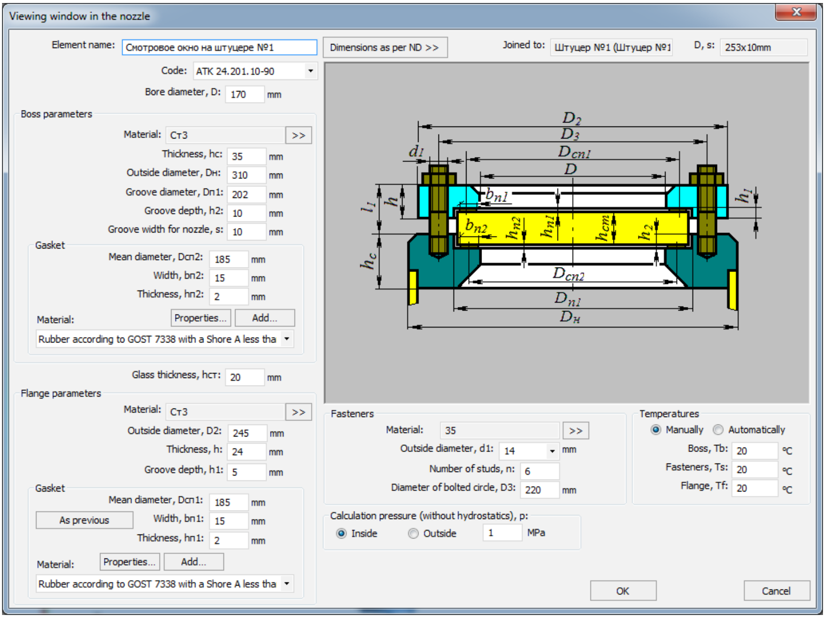


Fig. 3.208 Viewing window in the nozzle

This component acts in the structure similar to the head.Defining of properties is performed similar to the window on the weldolet.

3.17.67. Flange boss

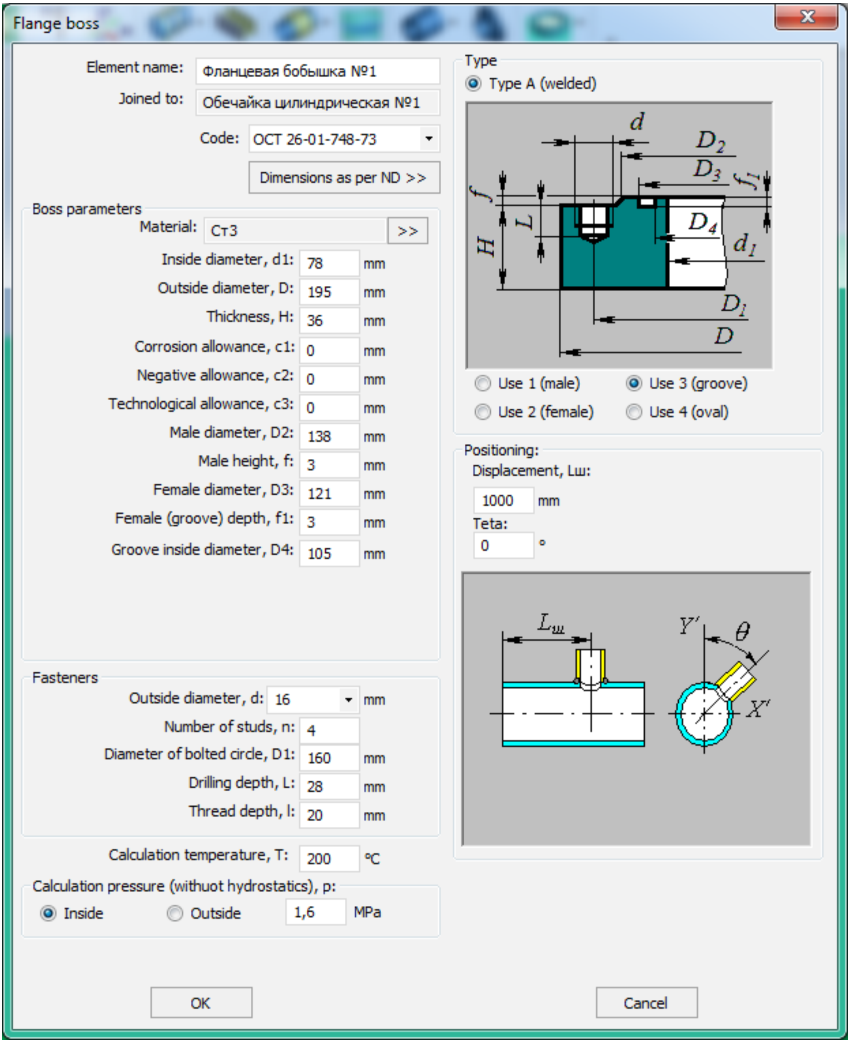
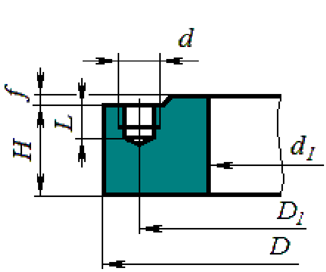
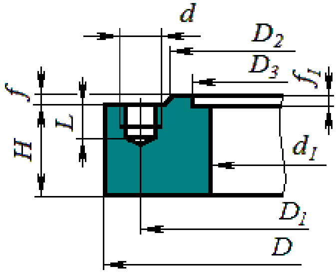


Fig. 3.209 Flange boss

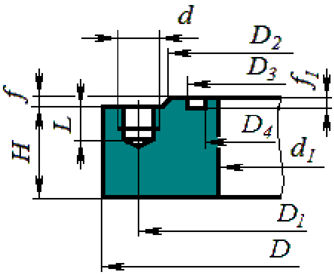
This component can be joined to cylindrical shell or elliptic head. Possible variants of structure:



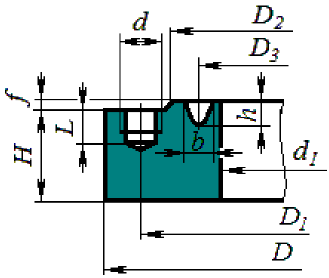
Male



Female



Groove



Oval gasket

Fig. 3.210 Flange boss types

3.17.68. Vessel assembly

This component provides creating of a model that has two or more vessels. (Fig. 3.211).

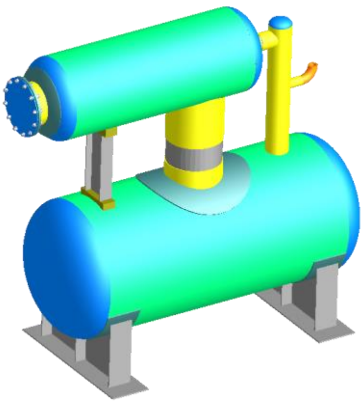


Fig. 3.211 Two-level vessel

This assembly is a coordinate system tied to some component, shifted and rotated relative to the source coordinate system (model CS or parent component CS).

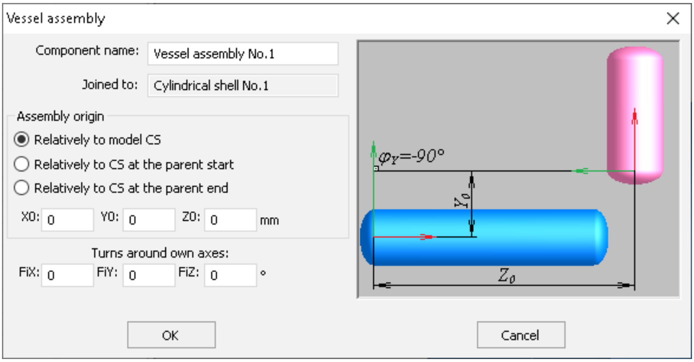


Fig. 3.212 Vessel assembly

New coordinate system is shifted relative to the old one at X_0, Y_0, Z_0, and then is rotated around its own axis X, Y, Z consequently to φ_X, φ_Y and φ_Z. Child components of the vessel, rotated in a specified manner, are joined to the assembly. To close the structure of the unit, the assembly shall be used together with the Rigid link component.

3.17.69. Rigid link

The component allows the two endpoints of the device to be rigidly connected at the level of a beam finite element model, by visually forming a rectilinear area of the given section.

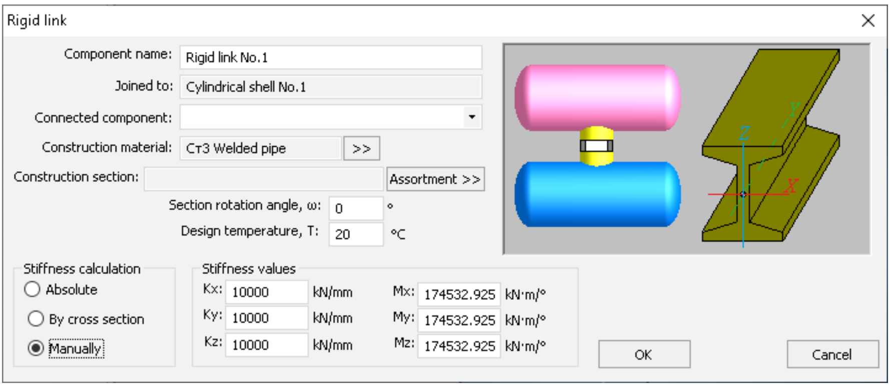


Fig. 3.213 Rigid link

The component is attached to the end of the shell or nozzle, or to the joining pad, or to the saddle support.

The other end area shall be selected from the list of available related components.

The weight of the material for this component is taken into account for a given section and is applied as a distributed weight load. The link is visually displayed with the specified section (if specified), and its stiffness is set by the "Stiffness calculation" option:

• Absolute - an absolutely rigid link is formed;

• By cross section - stiffness parameters are calculated for a beam element of a given section and length;

• Manually - the stiffness components must be entered.

3.17.70. Custom equipment

This component allows you to add arbitrary equipment created in a third-party CAD system to the model. These can be internal technological components, metal structures, external units, etc. The equipment will be visualized in the context of the model, as well as taken into account in the formation of the materials table and in the calculation of loads as a lumped mass rigidly connected to the selected parent component.

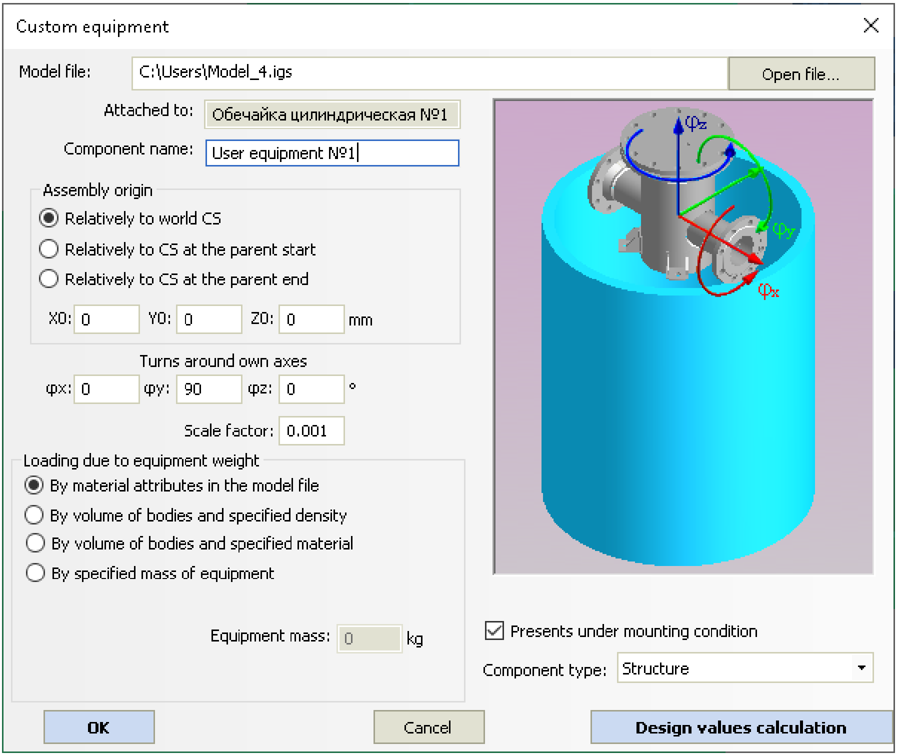


Fig. 3.214 Custom equipment

To load the equipment, it is necessary to prepare a file with its model in one of the popular data exchange formats and specify it in the “Model file” line. After loading, the assembly elements are stored in the vessel model (synchronization with the source file is not supported).

The options “Assembly origin”, “Turns around own axes” allow placing equipment at an arbitrary point of the model. The Scale factor option allows you to control the scaling of the custom model if the third-party CAD units are different from those used in the program.

The "Equipment weight load" option allows you to set the method by which the weight load will be calculated:

- “By material attributes in the model file” - if the assembly parts have a material density attribute set, the volume of each part is multiplied by the density;

- “By volume of bodies and specified density” - the value of the material density is set by user, the volume of each part is multiplied by the density;
- “By volume of bodies and specified material” – the material of equipment elements and its density is selected by user from the database, the volume of each part is multiplied by the density;
- “By specified mass of equipment” – the weight of equipment is set by user manually.

The “Component type” option (Equipment/Structure) is required for compatibility with the “Passat-Tanks” module:

- Loads from the weight of equipment and metal structures are calculated in different ways;
- Structures attached to the carcass roof are identified as carcass element.

3.17.71. Non-circular component

This component is intended for modeling and analysis of rectangular and oval structures.

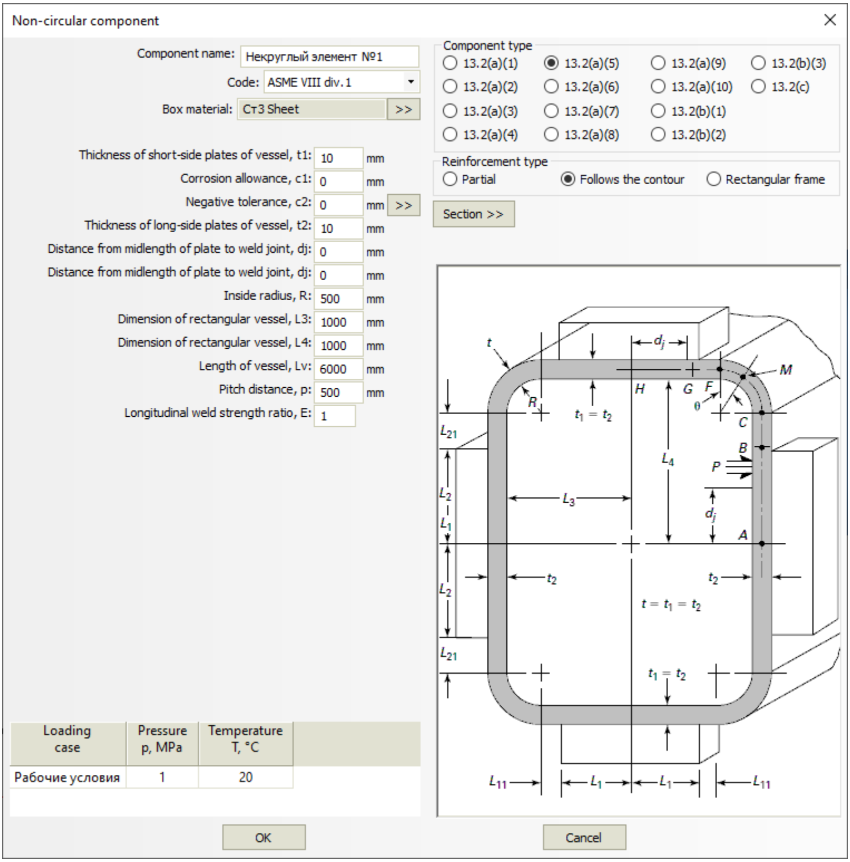


Fig. 3.215 Non-circular component

Available configurations of the component are determined by the code (Fig. 3.216).

Some configurations of the component can be reinforced with stiffeners (set by the “Section” button similar to the stiffening rings of cylindrical shells).

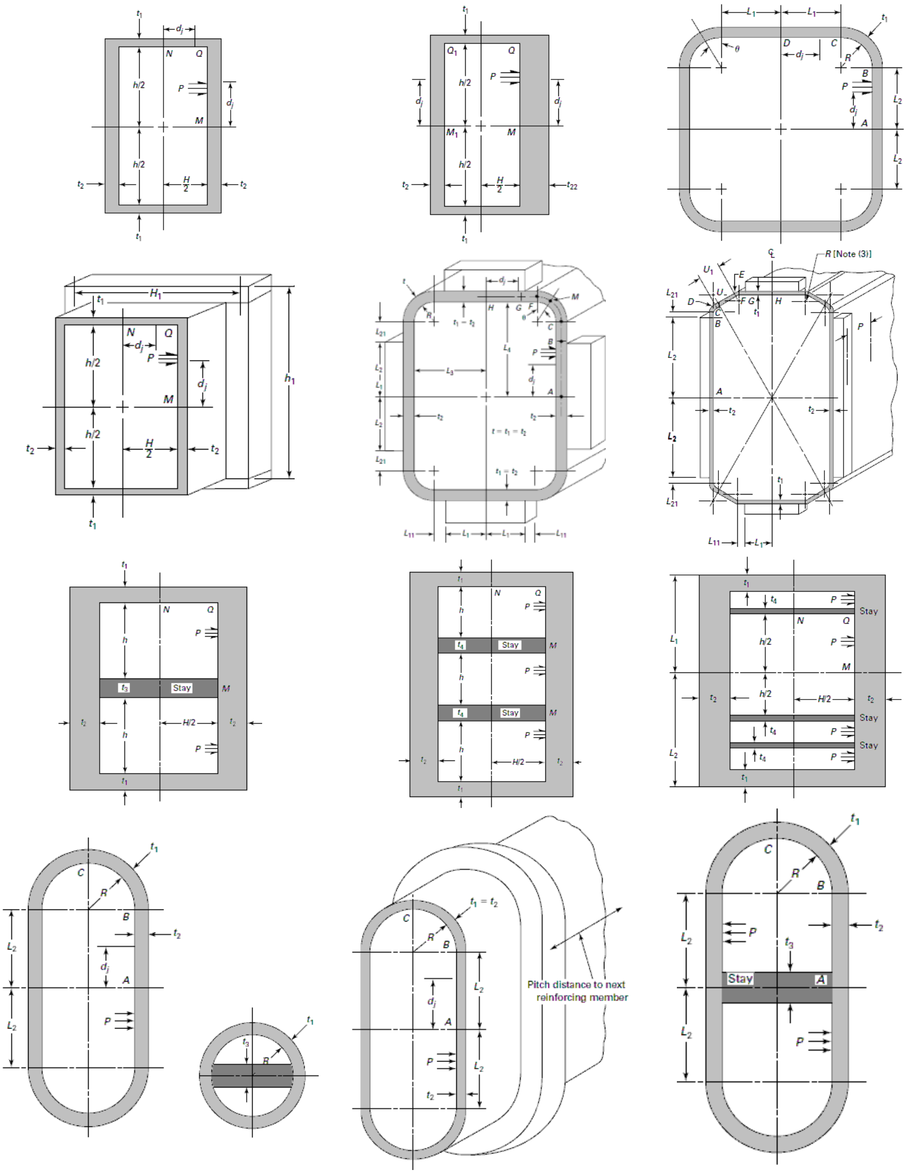


Fig. 3.216 Non-circular component as per ASME VIII-1

Currently, this component cannot be parent or adjacent to other design components.

3.18. Editing and deleting input data

After the model is created, its components can be edited or deleted, their colors can be changed, and existing components can be copied as new ones.

Select the desired component to edit or delete an component. If there are several components in one spot, select the desired component and choose the operation you wish to perform.

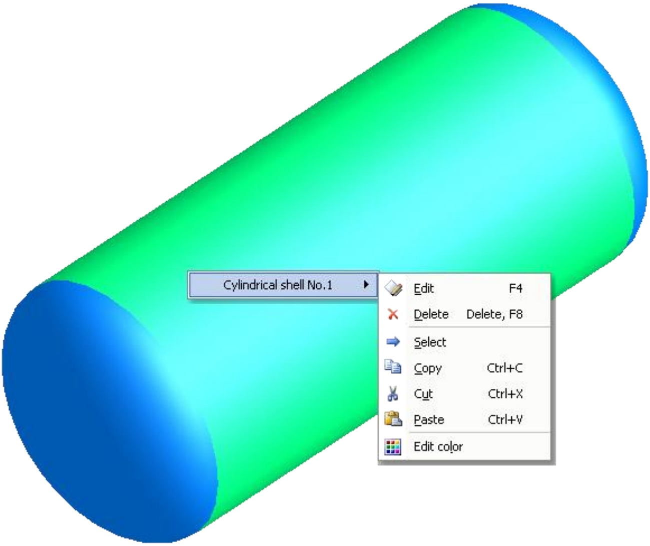


Fig. 3.217 Editing and deleting input data

Select whether to edit or delete the component. If component dimensions or load properties are changed, adjoining components of the whole model will also be changed (after a warning message is displayed).

Confirmation will be requested before deleting an component. In addition, if selected component includes daughter components (supports, nozzles, flanges, etc.), a warning message will be displayed before their deletion.

Components can also be edited or deleted by pressing "F4" or "F8", respectively, and choosing the desired component from a table.

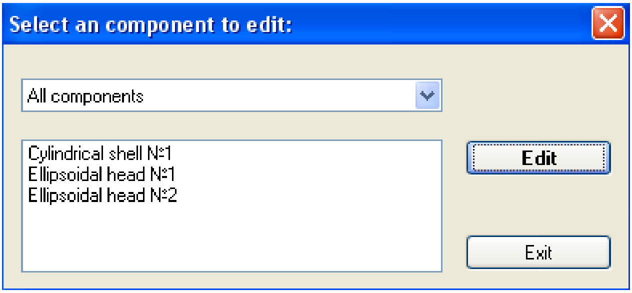
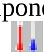


Fig. 3.218 Selection of the component

A component can be selected using the mouse cursor or the “Select” (icon) command of the pop-up menu (selected component will be highlighted). Selected component can be edited by double-clicking it or pressing “F4” and deleted by pressing “Delete” or “F8”. It can also be copied and then pasted. When pasting, the copied component will be adjoining to the selected one (if any). If no component is selected when pasting, a dialog with the list of possible adjoining components will appear.

3.18.1. Group data editing

If necessary to change some parameters (design temperatures, corrosion allowances) of several components within the model immediately, it can be conveniently done using icon  “Group data editing”.

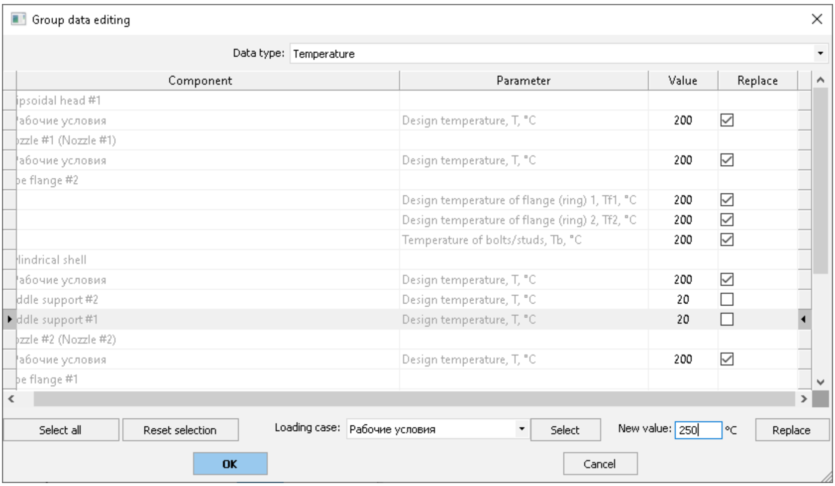


Fig. 3.219 Group data editing

In this window you can directly edit the available data cells, and also check a group of cells (buttons “Select all” and “Deselect” select/clear all the checks in the list). Then you can input a new value below and press “Replace” button. In the example at Fig. 3.219, all temperatures in the model, except for saddles supports ones, will be replaced from 200°C to 250°C.

3.18.2. Insulation setting by list

Insulation list” tool provides setting and changing of thermal insulation parameters of several model components.

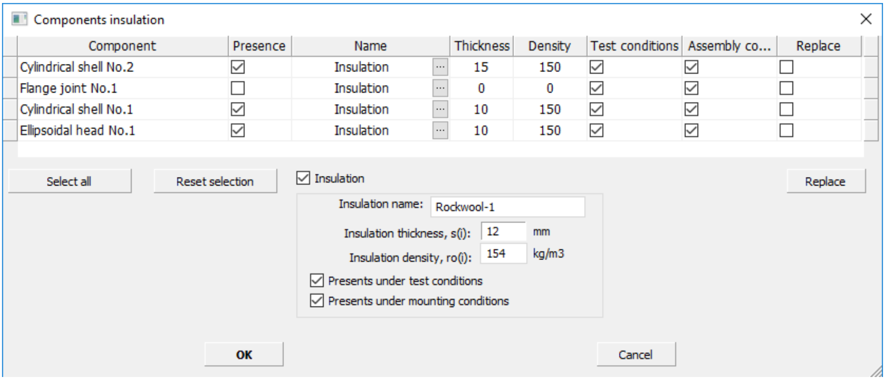


Fig. 3.220 Insulation setting by list

In the open window, you can directly edit the available cells, as well as check a group of cells (buttons “Select all” and “Deselect” select/clear all the checkmarks in the list). After that, you can enter the insulation sample parameters and click the "Replace" button. In the example shown in Fig. 3.220. after clicking the "Replace" button, thermal insulation on all components will be changed from mineral wool boards to fiberglass mats.

3.19. Data export and import

There are several import/export options in PASS/EQUIP. This is done by saving files in different formats. The following import/export options are currently available:

Format	**Description**
Export to XML **Import from XML**	Import/export to XML format. XML format data contain an object model and are sufficient for setting/retrieving all model properties required for vessel strength analyses. For more details, see Attachment “Passat XML”

Import from MechaniCS XML	
Export to Nozzle-FEM	When exporting to the "Nozzle FEM" program, the model is saved completely in XML format, while the target element is marked with a special tag, which allows the "Nozzle FEM" program to correctly interpret it. You must specify the folder where the exported files will be placed. The file names are the same as the components names. You must specify which of the components should be exported, or select an individual component using the context menu of the right mouse button (this can be a nozzle, a conical reducer, etc.).
Export of other file type to PASS/EQUIP file	When saving, a file type can be changed: i.e. a vertical model or column can be saved as horizontal model for calculations of tests in the horizontal position on the saddle supports.Not all of components can be saved in the new type of model, and appropriate notification will be displayed.
Export to IGES **Export to STEP** **Export to ACIS** **Export to Parasolid** **Export to** JT **Export to** VRML **Export to** STL	Creates the selected file format, containing geometric parameters of the vessel solid model and the attributes of object colors. In case simplified redesign mode is of switched on, the model will be additionally redesigned in precise mode, which may take more time. Obtained files can be opened and used for creation drawings of views and sections of the vessel in the popular 3D design and analysis systems (SolidWorks, Kompas-3D, ANSYS, etc.).
Export to Ansys (APDL)	The command is available for a vertical tank model (Passat-Tanks module). A finite element model of the structure is created, to which loads are applied based on the selected code (STO-SA-03-002, GOST 31385, API 650). The model is exported to an APDL batch file, which can be opened in Ansys to estimate the strength and buckling of the structure under a given loading mode. For details, see clause 3.19.1

When running a program with command line parameters **passat.exe File_Name /savexml**, the program saves an opened file in XML format in silent mode and ends.

3.19.1. Export of a tank model to Ansys

After creating a tank model, it can be exported to an Ansys batch file (APDL format) with constraints and applied loads, according to the selected code (Fig. 3.221, Fig. 3.222).

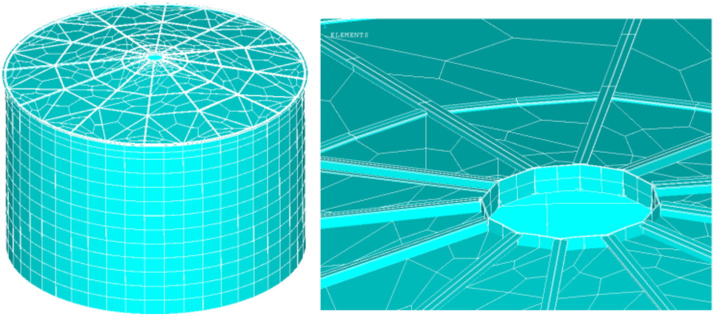


Fig. 3.221 Elements of the exported model

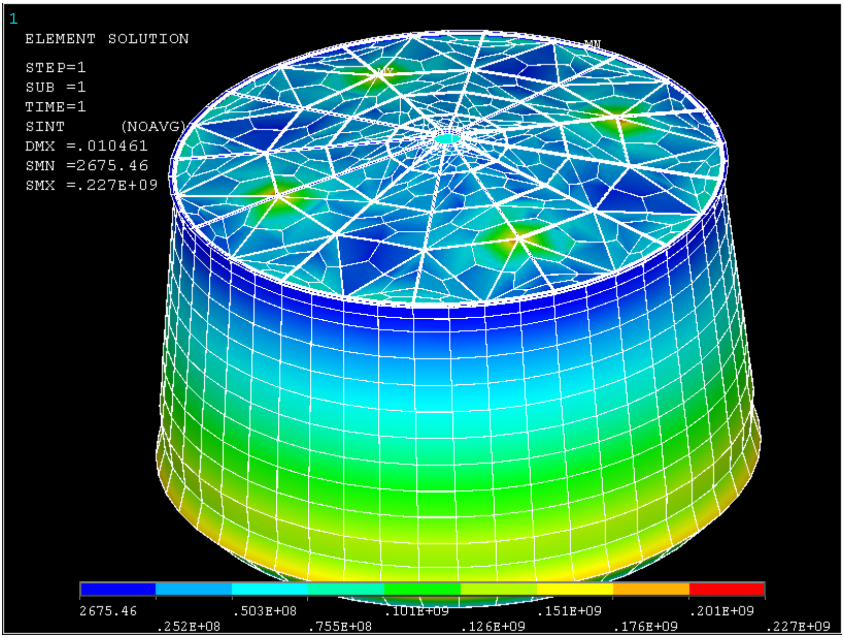


Fig. 3.222 Ansys exported model solution example (hydrostatic pressure loading, deformed view, stress intensity)

3.19.1.1 Model loading as per STO-SA-02-003, GOST 31385-2016

When selecting the option “Tank loading according to STO-SA-02-003, GOST 31385-2016” in the general data, the model is loaded according to the settings Fig. 3.223.

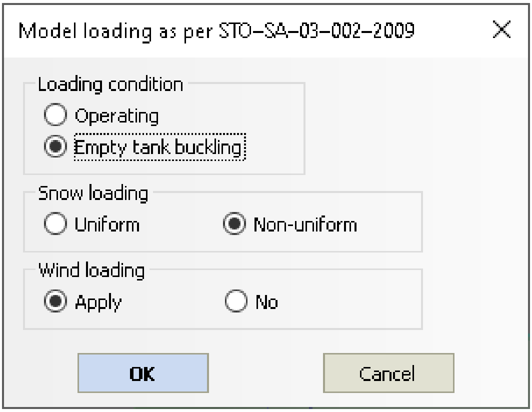
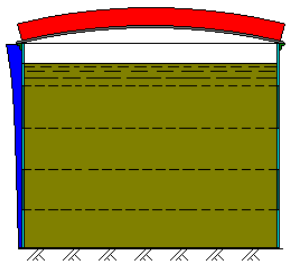
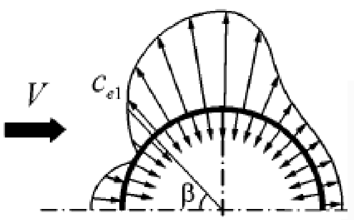
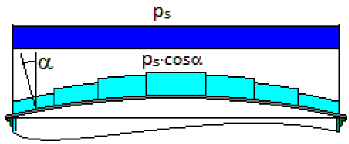
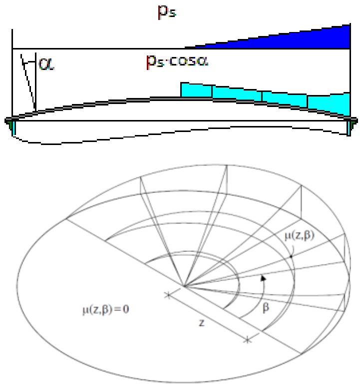
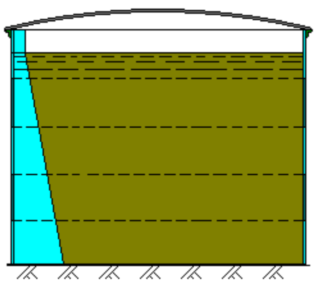


Fig. 3.223 Model loading as per STO-SA-02-003, GOST 31385-2016

Option	Description	Note
Wind loads	Wind loads, if specified, are applied to the roof (wind uplift as a constant internal pressure $0.9 \cdot p_w \cdot C_{e2}$ as per [35]) and to the wall (variable pressure diagram in height and in plan $0.9 \cdot k \cdot p_w \cdot C_{e1}$ as per [35])	 

Snow loads: uniform	The weight load due to snow is applied vertically downwards, taking into account the inclination of the normal to the roof surface: $p_s=0.9 \cdot p_{s0}$	
Snow loads: non-uniform	The weight load due to snow is applied vertically downwards, taking into account the inclination of the normal to the roof surface, as well as the unevenness factor as per [54]: $p_s=0.9 \cdot \mu \cdot p_{s0}$	
Operating	Loading with weight loads, internal vapor pressure and static head of the product: $p=\rho \cdot g \cdot (H-x)+1.2 \cdot p_i$ Wind loads are applied automatically (if any). The weight of the metal is taken in the corroded state. The weight of the attached metal structures and equipment is applied with a coefficient of 0.95. The equipment and metal structures on the wall are applied as a weight load distributed along the circumference.	

Empty buckling	External pressure on the wall: $p=0.95\cdot1.2\cdot p_v$ External pressure on the roof: $p=0.95\cdot1.2\cdot p_v + 0.95\cdot(1.05\cdot G_{r0}+1.3\cdot G_{rt})/(\pi\cdot r^2)$, where G_{rt} – roof insulation weight, G_{r0} – roof equipment weight. Snow and wind loads are applied optionally.	

3.19.1.2 Model loading as per API-650

When selecting the option "Tank loading according to API-650" in the general data, the model is loaded according to the settings Fig. 3.224.The minimum design load on the roof L_r=1.0 kPa is considered as a constant vertical load in the plan, it is applied to the roof elements taking into account the direction of the normal to the surface at the considered point ($L_r\cdot\cos\alpha$).

Uniform snow load $S_b=0.84\cdot S$, non-uniform snow load depends on roof slope: $S_u=\{S_b$ when $\theta\leq10°$; $1.5\cdot S_b$ when $\theta>10°\}$, is distributed over the roof sector 135° in the plan.

The design wind pressure on the shell $P_{WS}=0.89\cdot(V/190)^2$, is applied as a horizontal load from the windward side, taking into account the direction of the normal: $F_x=P_{WS}\cdot A_l\cdot\cos\alpha$, where A_l is the area of the considered wall element.

The design wind uplift pressure on the roof is $P_{WR}=1.48\cdot(V/190)^2$ applied as internal pressure to the roof elements (normal to the surface at the considered point).

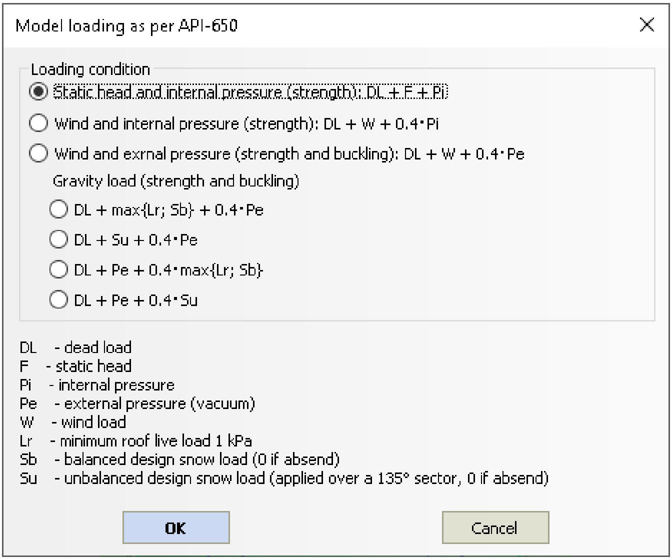


Fig. 3.224 Model loading as per API-650

3.20. Vessel components analysis and output of results

When model creation and input of load and material properties is complete, press **"Vessel analysis F3"** (or button) to run the analysis.

If vessel component dimensions and placement meet the analysis requirements, a detailed report containing strength analysis results and operability conclusions for each individual component will be displayed.

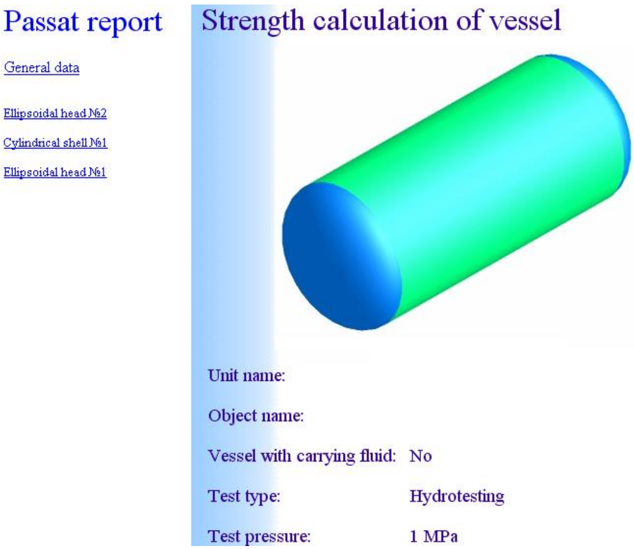


Fig. 3.225 Report window

3.21. Output in RTF format

Analysis output including intermediate and final results can be saved in RTF format. Output in RTF format is convenient in that its format can be set manually by using a template and that it can be edited in a word processor, such as Microsoft Word.

To export to RTF format, use the button (Ctrl+W). Output options can be set in the dialog window that will open (Fig. 3.226).

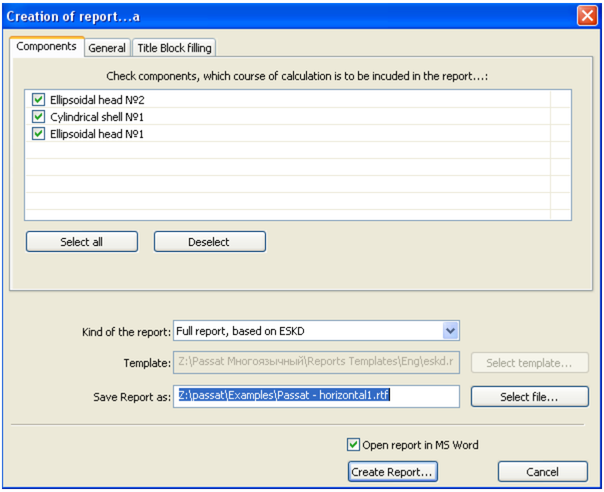


Fig. 3.226 Output in RTF format

Output options:

Output type	*Full report, based on ESKD* (Russian unified design documentation system) –contains input data and intermediate and final analysis results, including borders and title blocks in accordance with ESKD. *Brief report, based on ESKD* –contains input data and final analysis results, including bounds and title blocks in accordance with ESKD. It includes only criteria on which it is concluded about the vessel operability, and the values that they contain. *Full report in free format* –contains input data and intermediate and final analysis results. Output format is similar to the HTML format displayed at the end of analysis. *Based on user template* – output format is defined by the selected template.
Template	Allows selection of the template to be used for output.
Save report as	Allows input of the file name under which output should be saved.
Open output in MS Word	If selected, after output file is created, it will automatically be opened in MS Word (MS Word 2000

	or higher must be installed).
Create output	Generate output file.
Cancel	Close window without creating output file
"Elements" tab (Fig. 3.227**)**	Allow selection of model components to be included in output.
"General" tab (Fig. 3.227**)**	Allows input of vessel and plant names, analysis number and other values that can be included in the output. See 3.21.1.
"Title Block" tab	Allows input of information to be included in the Title Block of reports based on ESKD (or a custom format using a similar template)

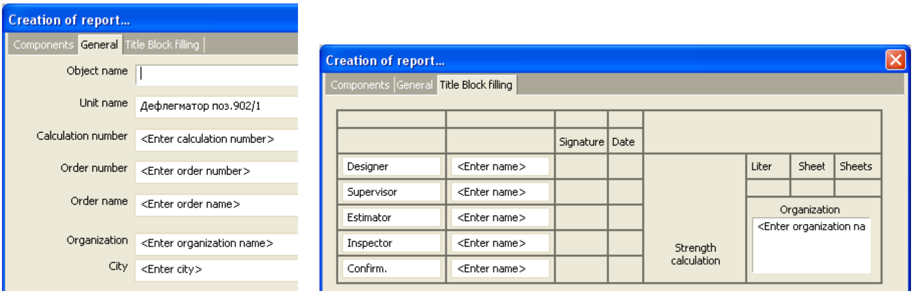


Fig. 3.227 Title block

3.21.1. Template creation

An output template must be in RTF format and can be created using any word processor supporting this format – for example, Microsoft Word. The template can include any components and formatting, which will be included the output. To insert appropriate data into the output, variables used by PASS/EQUIP must be included in the template. These variables will be substituted with actual values during output generation. A hatch sign (#) must be placed before a variable in the template. Two consecutive variables must be separated by a space. If a hatch sign not intended for variable definition must be placed, two consecutive hatch signs should be used (##).

Output of variables can be modified using parameters. Parameters are written inside the variable string, separated from the variable name by a colon and from each other by a comma. Parameters can have values, indicated by an equal sign.

#VARIABLE_NAME:PARAMETER[=VALUE],PARAMETER[=VALUE]…#

3.21.2. Use of variables

The following variables are currently available for use in templates:

Variable name	Description	Place of value
#OBJECT#	object name	"General" tab
#PLANT#	name of plant	"General" tab
#NCALC#	analysis number	"General" tab
#NORDER#	order number	"General" tab
#ORDER#	order number	"General" tab
#ORGANIZATION#	organization name	"General" tab, “Title Block” tab,
#CITY#	city of organization	"General" tab
#TITLE1# – #TITLE5#	job titles in Title Block text, top to bottom numbering	"Title Block" tab
#NAME1# – #NAME5#	surnames in Title Block text, top to bottom numbering	"Title Block" tab
#APPTITLE#	software name and version	set automatically
#PROGRESS#	vessel component analysis progress	set automatically in the range of the cycle **#<ELEMENT#**
#IMG#	figure containing vessel view	set automatically based on input options
#CALCDATE#	Analysis date and time	set automatically, according to current analysis time
#COMPLEX#	Determines whether the current parameter is a complex one (containing other parameters). For example, an component's material is a complex parameter.	set automatically in the range of the cycle effect **#<PARAMETERS#**. Set as «TRUE» or «FALSE»
#CALC#	Determines if the parameter is an intermediate analysis value	set automatically in the range of the cycle effect **#<PARAMETERS#**. Set as «TRUE» or «FALSE»
#NAME#	parameter name	set automatically in the range of the cycle effect **#<OPTIONS#**
#DIM#	parameter dimensions (if applicable)	set automatically in the range of the cycle effect **#<OPTIONS#**
#SYMB#	parameter symbol (if applicable)	set automatically in the range of the cycle effect **#<OPTIONS#**
#VAL#	parameter value	set automatically in the range of the cycle effect **#<OPTIONS#**
#RESULT#	Component analysis results (whether or not it meets the	set automatically in the range of the cycle effect

	standards)	**#<MODEL_COMPONENT#**. Set as «SUCCESS» or «FAIL».

To simplify the creation of a new template, an existing template installed with the software (eskd.rtf, located in the "Reports Templates" folder) can be used. Copy the template file under a new name and edit it.

Any variable can be used at any point in the template any number of times. Output variable text format will match formatting set in the template.

For example, if the template includes the following fragment:

Analysis was carried out by **"#ORGANIZATION#"**

#CITY#

where variable values are "NTP Truboprovod" and "Moscow" respectively, the output will look like this:

Analysis was carried out by **"NTP Truboprovod"**

Moscow

The only exception is the #REPORT# (#REPORT_BRIEF#) variable, the formatting of which is set automatically because it includes a large number of text fragments, titles, tables and figures describing the course of component analysis (full or brief).

3.21.3. Conditional variables

Conditional variables can also be used, which provide data output depending on fulfilment of various conditions. Conditional variable consists of two parts with a text fragment between them, which will be printed if conditions are met. The first part consists of text in the form of #<VARIABLE_NAME#. The second (closing) part consists of text in the form of #>VARIABLE_NAME#. Conditional variables can have dependant variables, the value of which is set automatically depending on the state of the conditional variable. The value of a conditional variable is output repeatedly until its value is false.

At the present time the following conditional variables are supported:

Variable name	Description	Dependant variables	Number of cycles
#<_IF_:condition# **#>_IF_:condition#**	True if ***condition*** is true. The	no	1

	condition can be a variable name (in which case the expression is true, if such variable exists) or a ***variable_name =value*** string.		
#<POURING#... #>POURING#	True, if vessel is used for filling **(**"Vessel carrying fluid" option in "General data" dialog box)	no	1
#<TEST# ... #>TEST#	True, if tests are performed ("Test type" option in "General data" dialog box)	No	1
#<ELEMENT#... #>ELEMENT#	True until all components (full or brief) from the list under the "Elements" tab are printed	**#REPORT#**	equal to the number of components included in the report
#<ELEMENT_BRIEF# ... #>ELEMENT_BRIEF#		**#REPORT_BRIEF#**	
#<MODEL_ELEMENT# ... #>MODEL_ELEMENT#	True until all components from the list under the "Elements" tab are printed, where components must meet additional parameters, if such parameters are set.	**#ELEMENT# #<PARAMETERS# #<ATTACHED#**	equal to the number of components included in the report
#<ATTACHED# #>ATTACHED#	True until all components adjoined to the current and adjoining components are printed	**#ELEMENT# #<PARAMETERS#**	equal to the number of components adjoined to the current and adjoining components
#<PARAMETERS# ... #>PARAMETERS#	True until all parameters of the current component are	**#<LEVEL# #COMPLEX# #CALC# #PIC# #NAME#**	equal to the number of component parameters

	printed.	**#DIM#** **#SYMB#** **#VAL#**	
#<LEVEL#... **#>LEVEL#**	Specifies parameter nesting level		equal to nesting level of the current parameter

Parameters of the #<MODEL_ELEMENT# conditional variable:

Parameter name	Description	Values	Default value
TOPLEVEL	"Top level" components, i.e. components composing the vessel shell	No	No
T	Component type	**CYL –** cylindrical shell **CYL_CLMN** – cylindrical shell of column **CONE** – conical shell **CONE_CLMN** – conical shell of column **NZL -** nozzle **ELL** – elllipsoidal head **CONEHEAD** – flat conical head **CONEHEAD_STEEP** – steep conical head **ELL_FLANGEAPP** – bolted elllipsoidal flange **FLANGEAPP** – vessel flange **FLANGEAPP_BOTT** – head flange **FLANGEAPP_ARM –** valve flange **FLAT_FLANGEAPP** – bolted flat head **FLATHEAD** – flat head **FLATHEADRIBS** – flat head with ribs **PACKING** – packing **RINGSTIFF** – stiffening ring **SADDLE** – saddle support **SKIRT** – supporting skirt **SPH** – spherical head **SPHBEADHEAD** – spherical head without knuckle **SPHBEAD_FLANGEAPP** – bolted spherical head **SUP_LUG** – supporting lugs **SUP_POLE**– supporting legs	No

For example, to display intermediate analysis data for all components, the following text can be entered in the template:

```
#<ELEMENT#
```

```
#REPORT#
#>ELEMENT#
```

Existing templates stored in the "Reports Templates" folder can be viewed as examples of template structure.

3.21.4. Embedding the vessel image

An image of the vessel can be inserted into the output using the #IMG# variable. The following parameters for the the #IMG# variable are available to adjust image output:

Parameter name	Description	Values	Default value
VIEW	view type	TOP – top view LEFT – left-side view FRONT – front view ISO – isometric view USER – custom view	FRONT
X	image width in pixels	1-65535	100
Y	image height in pixels	1-65535	100
STYLE	image style	SOLID – solid filling TRANSPARENT – transparent WIREFRAME – beam	SOLID
AA	if installed, perform image anti-aliasing (smoothing)	no	no

For example, to display an image containing beam top view of the vessel with dimension of 100x200 pixels, the following text must be entered:

```
#IMG:VIEW=TOP,X=100,Y=200,STYLE=WIREFRAME#
```

3.21.5. Embedding analysis time and date

Time and date can be inserted into the output in different formats using the #CALCDATE# variable. For time and date formatting, the #CALCDATE# variable has a string parameter DATEFORMAT, which can include the following fields:

Field	Description
%a	Abbreviated weekday name
%A	Full weekday name

%b	Abbreviated month name
%B	Full month name
%c	Date and time format appropriate for current locale
%d	Day of month as two digit number (01 – 31)
%H	Hour in 24-hour format (00 – 23)
%I	Hour in 12-hour format (01 – 12)
%j	Day of year as three digit number (001 – 366)
%m	Month as two digit number (01 – 12)
%M	Minute as two digit number (00 – 59)
%p	Current locale's A.M./P.M. indicator for 12-hour clock
%S	Second as two digit number (00 – 59)
%U	Week of year as two digit number, with Sunday as the first day of the week (00 – 53)
%w	Weekday as two digit number (0 – 6; Sunday is 0)
%W	Week of year as two digit number, with Monday as the first day of the week (00 – 53)
%x	Date format appropriate for current locale
%X	Time format appropriate for current locale
%y	Year without century, as two digit number (00 – 99)
%Y	Year with century, as four digit number
%z, %Z	Either time-zone name or time zone abbreviation, depending on registry settings; leave empty if time zone is unknown

For example, for date output in the DD.MM.YYYY HH:MM, format, the following text must be entered:

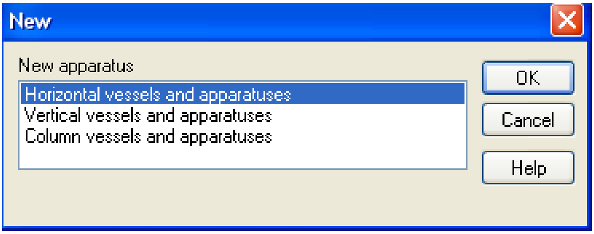
```
#CALCDATE:DATEFORMAT=%d.%m.%Y  %H:%M#
```

4. Example

4.1. Data input

An analysis of a horizontal vessel on saddle supports carrying petrochemical products (ρ=780 kg/m^3) with excessive internal pressure of 1 atm is given as an example. Excessive pressure during hydro-testing is 2 atm.

The vessel includes a shell, 5000mm in length and 2400mm in diameter, and two ellipsoidal heads. The vessel is placed on saddle supports, 300mm in width and with a wrapping angle of 120°, with reinforcing pads, 500mm in width, 12mm in thickness and with a wrapping angle of 140°. Corrosion allowance is 2mm. The vessel is under axial compression force of 100000N.



After entering vessel type and general data, model creation can begin.

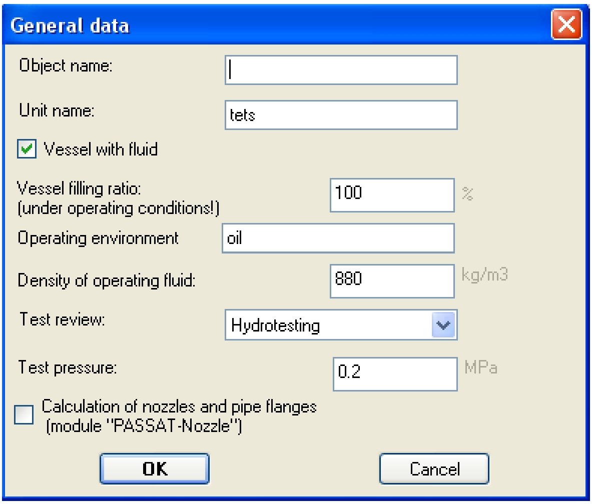
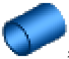
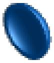


Fig.4.1 Example: general data

Dimensions, material properties and load properties for cylindrical shell , ellipsoidal heads  and saddle supports  are input.

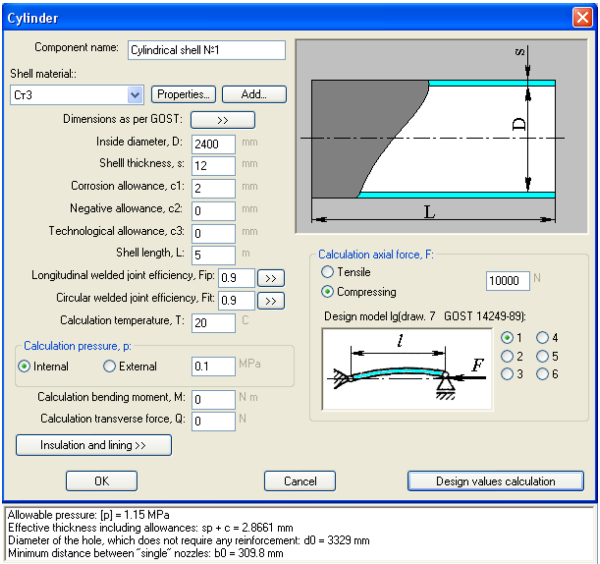


Fig.4.2 Example: shell

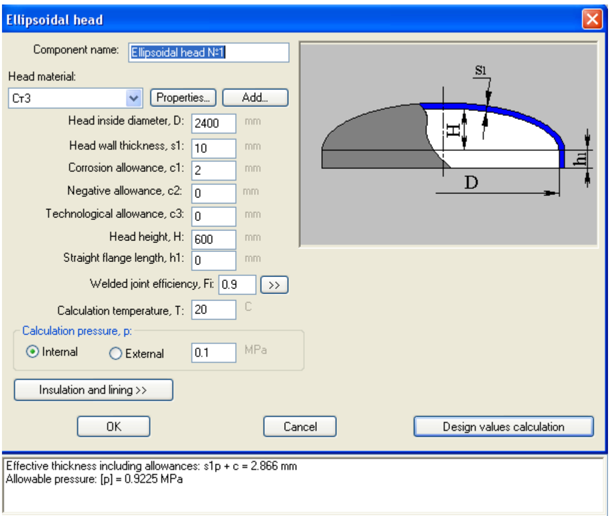


Fig.4.3 Example:head

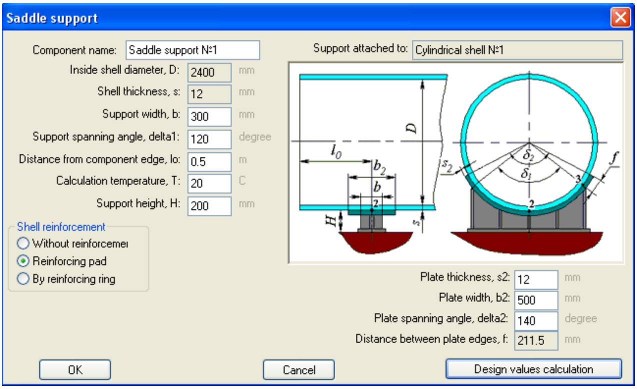


Fig.4.4 Example: saddle support

The following model will be displayed in the graphic display window.

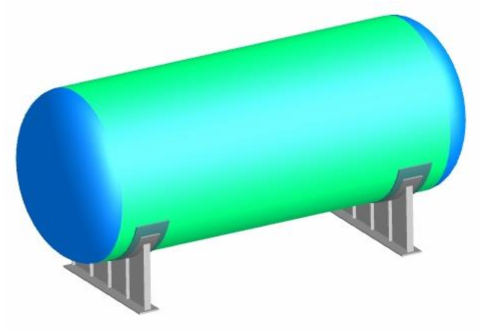


Fig.4.5. Example: calculation model

4.2. Analysis and output

To analyse the model, press **"Vessel analysis F3"** **(**or  button**).** In the course if analysis a detailed report with intermediate results is generated, which opens automatically upon the end of analysis. Ready report looks like a web page, the left part of which includes the table of contents (Fig.4.6).

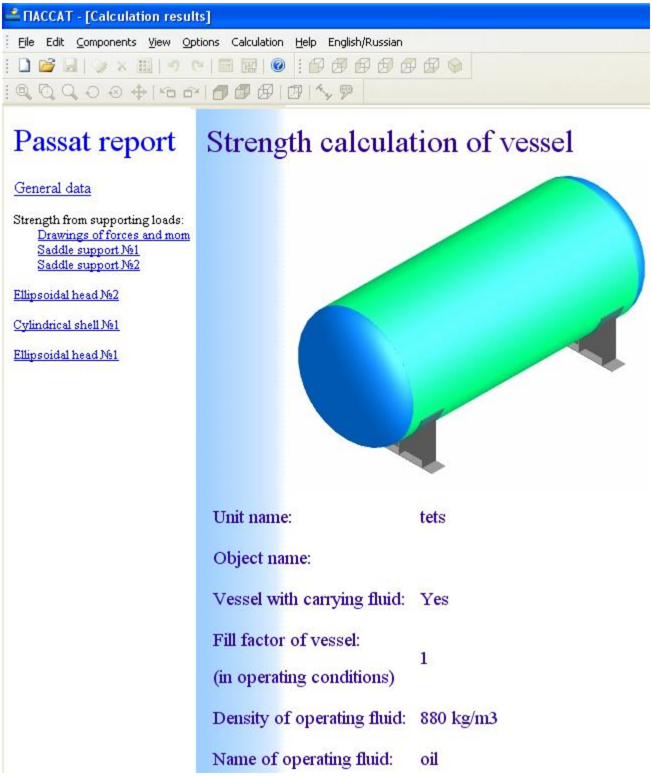


Fig.4.6 Example: report

To get detailed analysis output for a specific component, the component must be selected by clicking on it or pressing the "Tab" key.

5. References

1. SA 03-004-08. Strength Calculation of Vessels and Apparatuses. Norms and methods of strengthh calculation. Standard of Rostekhexpertiza Expert Association of technogenic hazardous facilities.
2. STO-SA-03.003-2009. Vessels and apparatuses. Norms and methods of strength calculation. Seismic loads calculation. Standard of Association of Expert organizations of hazardous facilities (Association Rostekhexpertiza)
3. GOST 34233.1-2017. Vessels and apparatuses. Norms and methods of strength calculation. General requirements.
4. GOST 34233.2-2017. Vessels and apparatuses. Norms and methods of strength calculation. Calculation of cylindrical and conical shells, dished and flat bottoms and heads.
5. GOST 34233.3-2017. Vessels and apparatuses. Norms and methods of strength calculation. Reinforcement of openings in shells and heads under external and internal pressure. Strength calculation of shells and heads under external static loads on the nozzle.
6. GOST 34233.4-2017. Vessels and apparatuses. Norms and methods of strength calculation. Strength and leak-tightness calculation of flange joints.
7. GOST 34233.5-2017. Vessels and apparatuses. Norms and methods of strength calculation. Calculation of the shells and heads from influence of support loads.
8. GOST 34233.6-2017. Vessels and apparatuses. Norms and methods of strength calculation. Strength calculation under low-cycle loads.
9. GOST 34233.7-2017. Vessels and apparatuses. Norms and methods of strength calculation. Heat exchangers.
10. GOST 34233.8-2017. Vessels and apparatuses. Norms and methods of strength calculation. Jacketed vessels and apparatuses.
11. GOST 34233.9-2017. Vessels and apparatuses. Norms and methods of strength calculation. Determination of stresses nozzle-to-shells and heads under action of pressure and external loads on the nozzle.
12. GOST 34233.10-2017. Vessels and apparatuses. Norms and methods of strength calculation. Vessels and apparatuses operating in hydrogen sulphide media.
13. GOST 34233.11-2017. Vessels and apparatuses. Norms and methods of strength calculation. Method of strength calculation of shells and head,

with provision for displacement of weld joint edges, angularity and out-of-roundness of the shells.

14. GOST 34233.12-2017. Vessels and apparatuses. Norms and methods of strength calculation. Requirements to the form of presentation of strength calculations made on computer.
15. GOST 14249-89. Vessels and apparatuses. Norms and methods of strength calculation.
16. GOST 25221-82. Vessels and apparatuses. Spherical heads and covers without knuckle. Norms and methods of strength calculation.
17. GOST 26202-84. Vessels and apparatuses. Norms and methods of strength calculation from influence of supporting loads on the shells and heads.
18. GOST 24755-89. Vessels and apparatuses. Norms and methods of strength calculation of holes reinforcement.
19. GOST 25859-83. Steel vessels and apparatuses. Norms and methods of strength calculation under low-cycle loads.
20. GOST R 51273-99. Vessels and apparatuses. Norms and methods of strength calculation. Determination of calculated forces against wind and seismic loads for column vessels.
21. GOST R 51274-99. Vessels and apparatuses. Column vessels. Norms and methods of strength calculation.
22. GOST 34283-2017. Vessels and apparatuses. Vessels and Apparatus. Norms and methods of strength calculation from wind loads, seismic influence and other external loads.
23. GOST 25867-83. Vessels and apparatuses. Jacketed vessels. Norms and methods of strength calculation.
24. GOST 30780-2002. Steel vessels and apparatuses. Bellows and sliding joints. Strength calculation methods.
25. GOST 26159-84. Iron vessels and apparatuses. Norms and methods of strength calculation. General.
26. GOST 27772-88. Mill products for constructional steel works.
27. GOST R 54522-2011. High pressure vessels and apparatus. Norms and methods of strength calculation.
28. GOST 26303-84 High-pressure vessels and apparatus. Threaded studs. Methods of strength calculations.
29. GOST R 55722-2013. Vessels and apparatus. Stress analysis code. Seismic analysis.

30. GOST 34283-2017. Vessels and apparatus. Norms and methods of strength calculation under wind loads, seismic influence and other external loads.
31. GOST 31385-2016. Vertical cylindrical steel tanks for oil and oil-products. General specifications.
32. OST 26-01-86-88. Fixed metal seals for vessels and equipment at pressures from 10 to 100 MPa.
33. OST 26-1046-87. High-pressure vessels and apparatus. Methods of strength calculations.
34. PNAE G-7-002-86. Strength calculation norms of the equipment and pipelines of atomic power plants. –M.: Energoatomizdat, 1989. – p. 525
35. SP 20.13330.2016. Set of rules. Loads and actions.
36. RD 24.200.08-90. Vessels and apparatuses. Norms and methods of stress analysis conical, ellipsoidal and spherical transitions.
37. RD 26-14-88. Vessels and apparatuses. Norms and methods of strength calculation. Components of heat exchangers.
38. RD 26-15-88. Vessels and apparatuses. Norms and methods of strength and leak integrity calculation of flange joints.
39. RD RTM 26-01-96-83. Flat round heads and heads with radial reinforcing ribs of vessels and apparatuses.
40. RD RTM 26-13-79. Heads and grids of air coolers. Strength analysis method.
41. RD 26-02-62-98. Strength calculation of vessel and apparatus components operating in corrosive hydrogen sulphide environments.
42. RD 26-02-63-87. Technical requirements towards designing and manufacturing of vessels, apparatuses and technological units of oil and gas treatment facilities working in environments causing hydrosulphuric corrosion cracking.
43. RD 10-249-98 Norms of strength calculation of stationary boilers and steam and hot-water pipelines.
44. OST 26-01-64-83. Fasteners. Construction and dimensions.
45. RD 26-01-169-89. Vessels and apparatuses. Norms and methods of strength calculation of the heads in attaching points of dumbbell piers.
46. RD 24.200.21-91. Vessels and apparatuses. Norms and methods of strength calculation of the components of floating heads within shell and tube heat exchangers.

47. RD 26-18-8-89. Weld joints of peep holes, nozzles and couplings welding. Basic types, structural components and dimensions.
48. RD 26.260.09-92. Vessels and apparatuses. Norms and methods of strength calculation of cylindrical shells and dished heads in junction points of nozzles under external static loads.
49. RTM 26-110-77. Strength calculation of cylindrical horizontal apparatus mounted on saddle supports.
50. RTM 26-111-77. Supports of cylindrical vertical vessels and apparatus. Norms and methods of strength calculation.
51. RD 26-01-149-84. Steel welded vessels and apparatuses with jackets made of semi-pipes located along generatrix. Codes and methods of strength calculation.
52. RD RTM 26-01-44-78. Details of pipelines for pressure over 10 to 100 MPa. Norms and methods of strength calculation.
53. SA 03-003-07. Strength and vibration calculation of steel technological pipelines. Standard of Rostekhexpertiza Expert Association of technogenic hazardous facilities.
54. STO-SA-03-002-2009. Rules for design, fabrication and assembly of vertical cylindrical steel tanks for oils. Standard of Rostekhexpertiza Expert Association of technogenic hazardous facilities.
55. Gorbachev M.V. "Heat and Mass Transfer". Publishing house of NSTU, 2015.
56. Standards of the Expansion Joint Manufactures Association, INC, 8th Edition, 2003.
57. EN 13445-3. European Standard. Unfired pressure vessels – Part 3. Issue 1 (2002-05).
58. EN 1991-1-4. Actions on structures - General actions - Wind actions
59. EN 1998-1. Design of structures for earthquake resistance. General rules, seismic actions and rules for buildings
60. ASME Boiler and Pressure Vessel Code. Sect.VIII, Div.1.
61. ASME VIII, Div 1, 2015. Rules for construction of pressure vessels.
62. ASME VIII, Div 2, 2015. Rules for construction of pressure vessels. Alternative rules.
63. ASME II, 2015.
64. WRC-107 Welding Research Council. Bulletin. “Local Stresses in Spherical and Cylindrical Shells due to External Loadings”. 1979.

65. WRC-297 Welding Research Council. Bulletin. “Local Stresses in Cylindrical Shells due to External Loadings on Nozzles – Supplement to WRC Bulletin №107”. 1987.
66. WRC-537 Welding Research Council. Bulletin. “Precision Equations and Enhanced diagrams for Local Stresses in Spherical and Cylindrical Shells due to External Loadings for implementation of WRC Bulletin 107. 2013.
67. BS-5500: 1976 Specification for Unfired fusion welded pressure vessels. British Standards Institution.
68. WRC-368 Welding Research Council. Bulletin. “Stresses in Intersecting Cylinders subjected to Pressure”. 1991. –32 p.
69. Bildy, Les M., 2000, “A Proposed Method for Finding Stress and Allowable Pressure in Cylinders with Radial Nozzles,” PVP Vol. 399, ASME, New York, NY, pp. 77-82.
70. Zick, L.P., “Stresses in Large Horizontal Cylindrical Pressure Vessels on Two Saddle Supports”, Welding Research Journal Supplement, September,1951.
71. Henry H.Bednar, Pressure Vessel Design Handbook. Second edition. 1986
72. Dennis R.Moss, Pressure Vessel Design Manual. 1987
73. AzDTN 2.3-1. Seysmik rayonlarda tikinti (zərbaycan Respublikası Dövlət Şəhərsalma və Arxitektura Komitəsi)
74. IS 1893. Indian Standard. Criteria for earthquake resistant design of structures.
75. API 650-2020. Welded Tanks for Oil Storage